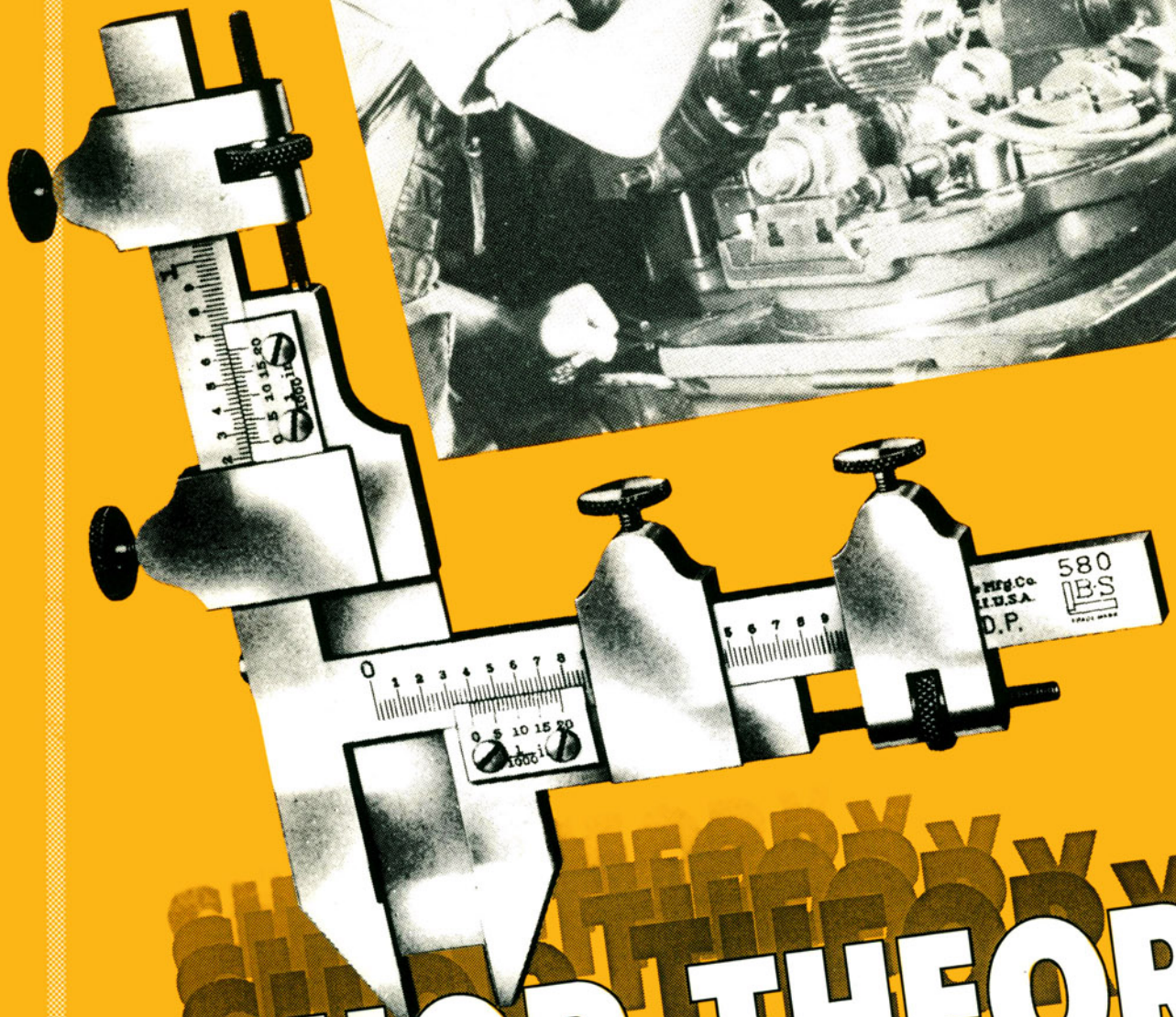
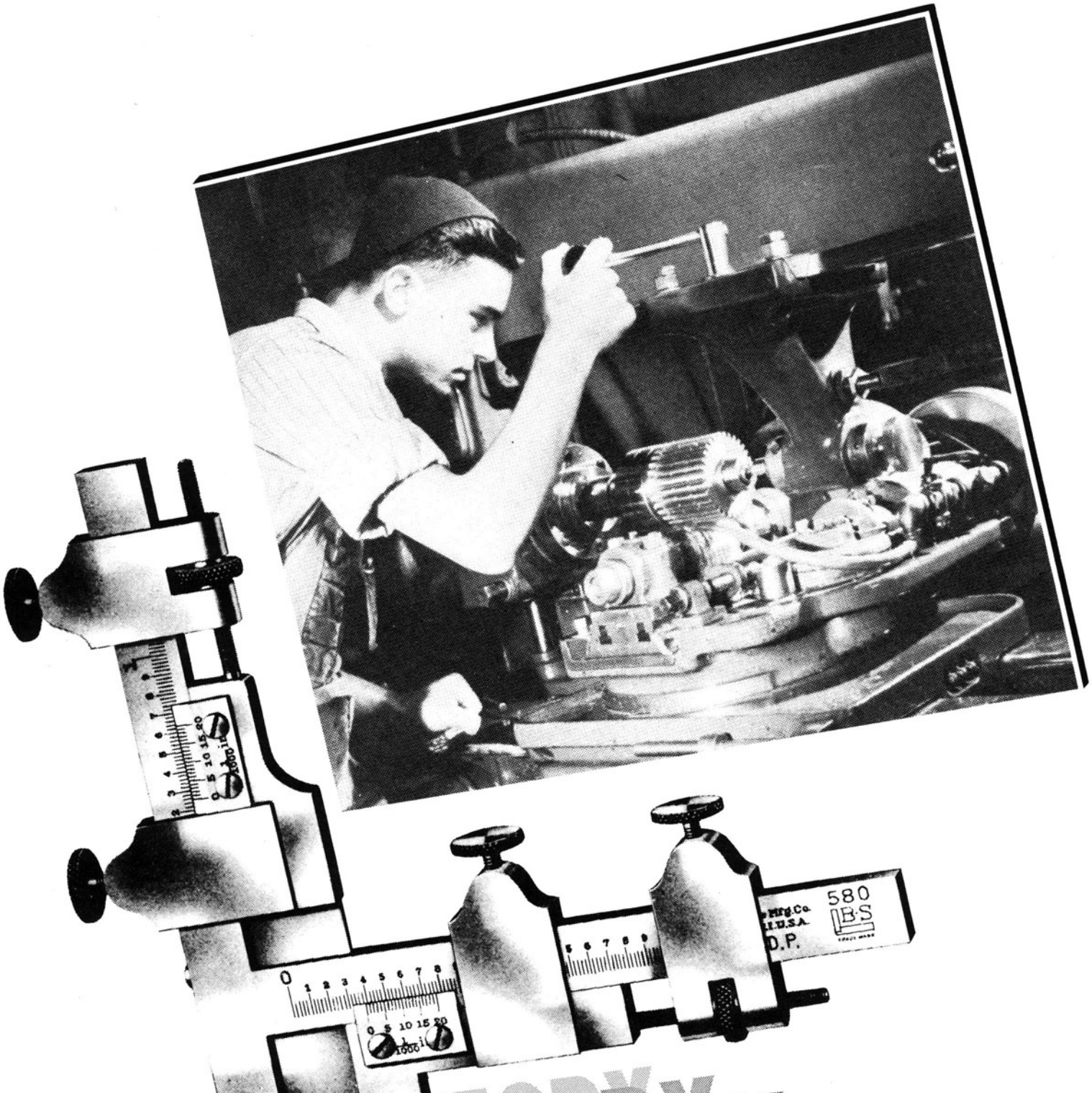


SHOP THEORY

PREPARED BY THE HENRY FORD TRADE SCHOOL, 1942

REPRINTED BY LINDSAY PUBLICATIONS INC



SHOP THEORY

PREPARED BY THE HENRY FORD TRADE SCHOOL, 1942

REPRINTED BY LINDSAY PUBLICATIONS INC

THIS PAGE IS BLANK

but this is not a printing or scanning
fault and no content is missing.

SHOP THEORY

Revised Edition

PREPARED BY

THE SHOP THEORY DEPARTMENT
HENRY FORD TRADE SCHOOL
DEARBORN, MICHIGAN

SHOP THEORY

Revised Edition

prepared by
The Shop Theory Department
Henry Ford Trade School
Dearborn, Michigan

Original Copyright 1934, 1941, 1942 by Henry Ford Trade School

Originally published by McGraw-Hill Book Co, New York

Reprinted by
Lindsay Publications Inc
Bradley IL 60915

All rights reserved.

ISBN 1-55918-006-4

5 6 7 8 9 0

WARNING

Remember that the materials and methods described here are from another era. Workers were less safety conscious then, and some methods may be downright dangerous. Be careful! Use good solid judgement in your work, and think ahead. Lindsay Publications Inc. has not tested these methods and materials and does not endorse them. Our job is merely to pass along to you information from another era. Safety is your responsibility.

Write for a complete catalog of unusual books available from:

Lindsay Publications Inc
PO Box 12
Bradley IL 60915-0012

PREFACE

From the beginning Shop Theory has played an important part in the educational program of Henry Ford Trade School. Because there was little material in print suited to our use, mimeographed sheets were prepared by our instructors. These were distributed in class and in time the student accumulated much information on many subjects.

Requests for these sheets were received from other schools and individuals in such numbers that we finally bound them in paper covers and sold them to those interested. More than 150,000 copies of previous editions have been furnished to high schools, colleges, industrial and vocational schools, United States Army and Navy schools, and many individuals in the United States and foreign countries.

During this process of developing the material now incorporated in this book many instructors contributed to make it a usable tool for student and teacher. Although it is impossible to give proper credit to all the instructors who helped bring the text to its present form, the following have contributed much and are entitled to recognition: Edward H. Bailey; Vincent C. Gourley, M. A.; Albert M. Wagener, M. E.; Roy E. Wipert; and John P. Heinz, who also did valuable work in rearranging and proofreading this edition.

We hope that because of the improvement in form and printing, this revised edition of Shop Theory may better meet the need of a larger number of students who are trying to fit themselves for the present emergency.

Frederick E. Searle, Superintendent
Ford Industrial Schools

ACKNOWLEDGMENTS

The Henry Ford Trade School gratefully acknowledges the courtesy of the following companies in permitting the use of various cuts and descriptive material from their textbooks and catalogues: Abrasive Machine Tool Company; Acme Machine Tool Company; Allen Manufacturing Company; American Swiss File and Tool Company; American Tool Works Company; Anderson Brothers Manufacturing Company; Armstrong Brothers Tool Company; Avey Drilling Machine Company; Baker Brothers, Inc.; Blanchard Machine Company; Bonney Forge and Tool Works; Boyar-Schultz Corporation; Brown Instrument Company; Brown & Sharpe Manufacturing Company; Bullard Company; Charles Bond Company; Carborundum Company; S. W. Card Manufacturing Company; Cincinnati Bickford Tool Company; Cincinnati Grinders, Inc.; Cincinnati Milling Machine Company; Cincinnati Planer Company; Cleveland Twist Drill Company; Desmond-Stephan Manufacturing Company; Henry Disston and Sons, Inc.; Detroit Torch and Manufacturing Company; Eugene Dietzgen Company; Ex-Cell-O Corporation; Farrel Birmingham Company, Inc.; Fellows Gear Shaper Company; Foster Machine Company; Frederick Post Company; Gleason Works; Goodell-Pratt Company; Gould and Eberhardt; Greenfield Tap and Die Corporation; Heald Machine Company; John Bath and Company, Inc.; Jones & Lamson Machine Company; Kearney and Trecker Corporation; Landis Tool Company; Leeds and Northrup Company; Lucas Machine Tool Company; Lodge and Shipley Machine Tool Company; Lufkin Rule Company; Millers Falls Company; Moline Tool Company; Monarch Machine Tool Company; Morse Twist Drill and Machine Company; National Twist Drill and Tool Company; Nicholson File Company; Norton Company; Oliver Instrument Company; Pratt and Whitney Aircraft; Ransom Grinding Machine Company; Reed Manufacturing Company; Reed-Prentice Corporation; Rivett Lathe and Grinder, Inc.; Simonds Saw and Steel Company; Skinner Chuck Company; South Bend Lathe Works; Standard Fuel Engineering Company; Stanley Rule and Level Division of the Stanley Works; Swann Chemical Company; Charles A. Strelinger Company; J. T. Slocomb Company; L. A. Sayre Company; L. S. Starrett Company; Taft-Peirce Manufacturing Company; Taylor and Fenn Company; Union Tool Company; Walton Company; Warnock Manufacturing Company; Warner and Swasey Company; O. S. Walker Company, Inc.; J. H. Williams Company; Western Tool and Manufacturing Company; Whitman and Barnes; Wilson Mechanical Instrument Company, Inc.

TABLE OF CONTENTS

Page

PREFACE v

ACKNOWLEDGMENTS vi

Chapter 1 - DECIMAL EQUIVALENTS 1

Chapter 2 - FORMULAS 3

Chapter 3 - SMALL TOOLS 5

Hammers--Hack saws--Chisels--Wrenches--Screw drivers--Screws--Drills--Countersinks--Counterbores--Reamers--Taps--Dies--Gages--Rules--Clamps and devices to hold work--Surface plate--Straight edge--Tool and work holders--Arbors--Cutters--Scrapers--Micrometers--Sine bar--Calipers--Dial indicators--Protractors--Toolmakers' buttons--Bench centers--Soldering tools--Carpenters tools--Dividers--Trammel.

Chapter 4 - RULES 33

Rules--Scales--Kinds of rules--Graduations.

Chapter 5 - MICROMETERS 37

Types--Principal parts--How to read--Reading in ten-thousandths--Screw thread micrometer.

Chapter 6 - VERNIER GAGES 41

Height gage--Vernier caliper--Gear tooth vernier caliper--Vernier bevel protractor.

Chapter 7 - CHISELS AND CHIPPING 45

Definitions--Chipping by hand--Angle on the cutting edge--Uses of different types of chisels--Hints on chipping.

Chapter 8 - HACK SAWS AND SAWING 48

Hand hack saw blade--Placing in a frame--Number of strokes per minute--"Set" of a saw--When to use different kinds of blades--Number of teeth per inch--Pressure required--Effect of using a coolant--Causes for breaking of blades--Slotting hack saw blades--Length measurement.

Chapter 9 - FILES AND FILING 52

Kinds of files--Classifications--Use--Pitch--Length--Shapes--How to hold a file--Draw filing--Pinning--Cleaning--Filing on the lathe--Precautions when filing--Filing machine.

Page

Chapter 10 - SOLDERING..... 59

Definition--Composition of solder--Flux, kinds for various metals--Muriatic acid, "killed" raw state--How to "tin" a soldering copper--Preparing a surface--How to solder cast iron, aluminum--"Sweating" --Hard and soft solder--Rosin--Electrical connections--Tendency of solder to run--Brazing--Adding bismuth, cadmium, zinc, aluminum, phosphorus to solder--Tip design and proper use of square point soldering copper.

Chapter 11 - SHOP REVIEW..... 62

Measurements--Decimals--Fractions--Square root--Circle--Quadrilateral--Triangle--Threads per inch on micrometer screw--Prick punch and center punch--Screw driver--Cold chisel--Drills--Files--American National threads--Stating dimensions when ordering stock--Brass--Steel, type designations--Types and sizes of screws.

Chapter 12 - DRILLS AND DRILLING OPERATIONS..... 69

Twist drills--Grinding--Oil hole drills--Flat drills--Starting the drill--Laying out the holes--Drawing back on center--Coolant--Cutting speeds--Tapping--Reaming--Lapping--Countersinking--Counterboring--Center drilling--Spot facing--Plain vertical spindle drilling machine--Sensitive drilling machine --Multiple spindle drilling machine--Radial drilling machine--Decimal equivalents of drill sizes--Table of cutting speeds--Tap drill sizes.

Chapter 13 - TAPERS..... 84

Definition--Used on drills, milling machines, lathe centers--Jarno system--Checking gages--Exercises to determine the taper per foot--Angle to set taper attachment--Taper plugs, determining diameter and TPF --Taper dowel pin--Producing tapers on a lathe--Taper hole produced with a taper reamer.

Chapter 14 - THREADS..... 90

Terms relating to screw threads--Terms relating to classification and tolerance--American National form of threads--American National Standard thread--Sharp "V" thread--Acme thread--Brown and Sharpe worm thread --Square thread--Buttress thread--Kinds of taps--Dies--Size of hole to drill to tap--Size of blank stock--Lubricant--Setting the compound rest for cutting screw threads--Setting the tool to cut pipe threads--Some methods of producing a thread--Threads used on automobile parts--Right-hand and left-hand threads--Taper pipe thread notation and dimensions--Gaging pipe threads--Thread micrometer--Table of coarse and fine thread series--Three-wire system of checking the pitch diameter of screw threads--Commercial size of wire to check threads--Bench comparator--Nut threads.

Page

Chapter 15 - GEARING..... 101

Spur gearing--Definitions of terms--Calculating dimensions, problems--Methods of producing gears--Gear cutters--Gear generator--Questions on gearing.

Chapter 16 - CUTTING TOOLS..... 112

Material--Use--Grinding, rake and clearance--Setting cutting edge--Shapes of turning tools--Grinding boring tools--Shaper and planer tool bits.

Chapter 17 - SHAPER..... 115

Classes, crank and geared--Size--Difference between a plain and a universal shaper--Operating--Shaper and planer tool bits--Shaping a rectangular job--Chattering--Vertical and angular cuts--Cutting and checking a dovetail--Method used to shape a "V" or keyway--Slotter.

Chapter 18 - PLANER..... 125

Size--Planer tools--Chuck--Centers--Clamps, toe-dogs, and poppets--Hints on clamping work to table.

Chapter 19 - LATHES..... 130

Types--Determining size--Important parts and their functions--Compound rest--Attachments--Cutting tools--Placing tool holder in tool post--Dead and live centers--Mounting work between centers--Setting and using the tool--Check work with drawing--Boring end of shaft--Operations performed on the lathe--Allowance for grinding stock--Under-cutting --Cutting speeds--Producing tapers--Center gage--Cutting threads--Knurling--Safety rules.

Chapter 20 - TURRET LATHES..... 143

Horizontal universal turret lathe--Full universal threading turret lathe--Permanent setup for bar and chucking work--Stationary overhead pilot bar--Vertical turret lathe.

Chapter 21 - MILLING MACHINE..... 156

Classes--Attachments--Index--Vise--Chuck--Spindle --Cutters--Operations--Setting cutter central with work--Dividing head--Indexing, rapid, plain, and differential--Index tables--Graduating--Angular indexing Table, use--Angular indexing--Helical and spiral milling--Short lead milling--Cam cutting attachment--Cutter speeds--Table of cutting speeds.

Chapter 22 - GAGES AND GAGE BLOCKS..... 185

Types of gages--Sine bar--Jo-blocks--Amplifying comparator--Fitting a shaft to a hole--Precision measurement--Johansson gage blocks--Finding the size of blocks to be used--Building the combination--Applications of blocks and accessories--Review and problems.

Page

Chapter 23 - HEAT TREATMENT..... 197

Steel, classifications and designations of types--Operations in heat treatment--Critical points--Furnaces--Hardening--Cyaniding and carburizing--Nitriding--Percentage of carbon--Steel for tool bits--Determining hardness of steel, Rockwell, Brinell, Scleroscope--Suggestions in heat treatment--Safety rules in heat treating steel--Steel analysis chart--Spark test chart--Heat treatment chart--Hardness conversion table.

Chapter 24 - ABRASIVES AND GRINDING WHEELS..... 209

History--Characteristics and manufacture--Grinding wheels--Bonds--Wheel structure--Wheel grading--Wheel selection--Grinding theory--Grinding wheel speeds--Chatter or vibration--Glazing and loading--Dressing--Rough grinding--Care of grinding wheels--Table for determining speeds of various sizes of wheels--Questions for review.

Chapter 25 - GRINDING MACHINES..... 216

Definitions--Types--Cutter grinder--Sharpening of cutters--Tooth rests--Sharpening side teeth of milling cutter--Important facts to remember when sharpening cutters--Safety rules for cutter grinders--Questions--Cylindrical (external) grinding--Steady rest--Feeds--Work speeds--Operation errors--Plunge cut grinding--Shoulder work--Gaging methods--Taper work--Wheel dressers--Safety items--Universal grinders--Centerless grinders--Methods of feeding--Safety rules for external grinders--Questions--Surface grinders--Planer type--Rotary type--Surface grinder wheels--Mounting surface grinder wheels--Safety flanges--Dressing surface grinder wheels--Holding work--Witness marks--Angle plate work--Grinding work square--Job analysis--Groove and slot grinding--To use the radius circle--Horizontal rotary surface grinder--Vertical spindle rotary grinder--Tools commonly used on surface grinders--Safety rules for surface grinders--Questions--Internal grinding--Universal internal grinder--Plain internal grinder--Work rotating type--Methods of holding work--Extreme taper--Bell-mouthing--Blind holes--Internal centerless grinder--Cylinder or planetary grinder--Tools commonly used on internal grinders--Common set-ups and operations on internal grinders--Safety rules for internal grinding--Questions--Miscellaneous grinders --Grinding suggestions --Precision thread grinder.

Chapter 26 - ROUTING OF BENCH TOOL WORK..... 264

Term "bench work"--Routing of the job--An example of the operations and routing of a job--Blueprints--Order form--Detail, work card--Tool operations card.

Chapter 1

DECIMAL EQUIVALENTS

The measurements made in a machine shop are usually taken in inches or fractional parts of an inch. Most of the precision tools used in the shop read in thousandths of an inch. The usual graduations on a rule are in 64ths, 32nds, 16ths, and 8ths of an inch.

Before a student can read a rule or the precision measuring tools efficiently, he must be thoroughly familiar with fractions and decimal fractions. Since he is often called upon to change decimals to fractions and fractions to decimals in making measurements, and in reading and checking blue prints and sketches, he should understand this operation thoroughly. Precision measuring tools, such as micrometers and vernier tools, are read in thousandths or fractional parts of a thousandth of an inch. For example, 1/16 is read sixty-two and one-half thousandths, 1/32 is read thirty-one and one-quarter thousandths, 1/64 is read fifteen and five-eighths thousandths, etc. It will be noted that these readings give the full decimal values for the corresponding fractions. However, since the precision tools commonly used in the shop cannot be read closer than one-tenth of one thousandth, it is customary for a mechanic to use only those figures up to and including the tenth-thousandth figure, or to four decimal places. For example, the complete decimal value of 1/64 is .015625, which in the shop is commonly read as fifteen and six-tenths thousandths, fifteen being the whole number of thousandths while six is six-tenths of one one-thousandth, or a fractional part of a thousandth.

To change a fraction to a decimal, divide the numerator by the denominator. For example, in changing 3/16 to a decimal, 3.0000 ÷ 16 = .1875.

When reading a rule it is sometimes convenient to read either way from some large dimension line. That is, in measuring 47/64 of an inch it is easier to find 3/4 and subtract 1/64 from it than to count the divisions from the end of the rule. The following table contains the decimal values and fractional values in halves, fourths, eighths, sixteenths, thirty-seconds, and sixty-fourths, for the fractions most commonly used in the shop. These values should be memorized.

Read the decimal equivalent to the

1/64	-	-	-	-	-	.015625	7/16	28	14	7	-	-	.4375
1/32	2	-	-	-	-	.03125	1/2	32	16	8	4	2	.500
1/16	4	2	-	-	-	.0625	9/16	36	18	9	-	-	.5625
3/32	6	3	-	-	-	.09375	5/8	40	20	10	5	-	.625
1/8	8	4	2	1	-	.125	11/16	44	22	11	-	-	.6875
3/16	12	6	3	-	-	.1875	3/4	48	24	12	6	3	.750
1/4	16	8	4	2	1	.250	13/16	52	26	13	-	-	.8125
5/16	20	10	5	-	-	.3125	7/8	56	28	14	7	-	.875
3/8	24	12	6	3	-	.375	15/16	60	30	15	-	-	.9375
	64	32	16	8	4		1	64	32	16	8	4	1.0000

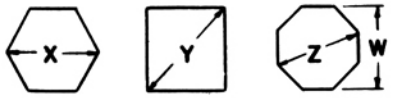
fourth decimal place and express in thousandths. For example, 23/64 is read three hundred fifty-nine and four tenths, thousandths.

Memorize the quarters, eighths, sixteenths, one thirty-second, three thirty-seconds, and one sixty-fourth.

Fractions	64ths	32nds	16ths	8ths	4ths	Decimal Equivalents	Fractions	64ths	32nds	16ths	8ths	4ths	Decimal Equivalents
1/64	-	-	-	-	-	.015625	33/64	-	-	-	-	-	.515625
1/32	2	-	-	-	-	.03125	17/32	34	-	-	-	-	.53125
3/64	-	-	-	-	-	.046875	35/64	-	-	-	-	-	.546875
1/16	4	2	-	-	-	.0625	9/16	36	18	-	-	-	.5625
5/64	-	-	-	-	-	.078125	37/64	-	-	-	-	-	.578125
3/32	6	-	-	-	-	.09375	19/32	38	-	-	-	-	.59375
7/64	-	-	-	-	-	.109375	39/64	-	-	-	-	-	.609375
1/8	8	4	2	-	-	.125	5/8	40	20	10	-	-	.625
9/64	-	-	-	-	-	.140625	41/64	-	-	-	-	-	.640625
5/32	10	-	-	-	-	.15625	21/32	42	-	-	-	-	.65625
11/64	-	-	-	-	-	.171875	43/64	-	-	-	-	-	.671875
3/16	12	6	-	-	-	.1875	11/16	44	22	-	-	-	.6875
13/64	-	-	-	-	-	.203125	45/64	-	-	-	-	-	.703125
7/32	14	-	-	-	-	.21875	23/32	46	-	-	-	-	.71875
15/64	-	-	-	-	-	.234375	47/64	-	-	-	-	-	.734375
1/4	16	8	4	2	-	.250	3/4	48	24	12	6	-	.750
17/64	-	-	-	-	-	.265625	49/64	-	-	-	-	-	.765625
9/32	18	-	-	-	-	.28125	25/32	50	-	-	-	-	.78125
19/64	-	-	-	-	-	.296875	51/64	-	-	-	-	-	.796875
5/16	20	10	-	-	-	.3125	13/16	52	26	-	-	-	.8125
21/64	-	-	-	-	-	.328125	53/64	-	-	-	-	-	.828125
11/32	22	-	-	-	-	.34375	27/32	54	-	-	-	-	.84375
23/64	-	-	-	-	-	.359375	55/64	-	-	-	-	-	.859375
3/8	24	12	6	-	-	.375	7/8	56	28	14	-	-	.875
25/64	-	-	-	-	-	.390625	57/64	-	-	-	-	-	.890625
13/32	26	-	-	-	-	.40625	29/32	58	-	-	-	-	.90625
27/64	-	-	-	-	-	.421875	59/64	-	-	-	-	-	.921875
7/16	28	14	-	-	-	.4375	15/16	60	30	-	-	-	.9375
29/64	-	-	-	-	-	.453125	61/64	-	-	-	-	-	.953125
15/32	30	-	-	-	-	.46875	31/32	62	-	-	-	-	.96875
31/64	-	-	-	-	-	.484375	63/64	-	-	-	-	-	.984375
1/2	32	16	8	4	2	.500	1 inch	64	32	16	8	4	1.000

FORMULAS

Fig. 1



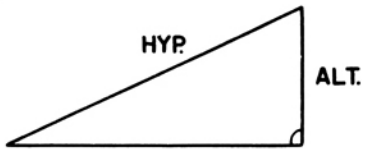
W= WIDTH

X = 1.1547. W

Y = 1.4142.W

Z = 1.0824.W

Fig. 2



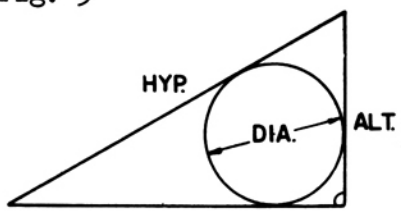
BASE

$HYP. = \sqrt{BASE^2 + ALT.^2}$

$BASE = \sqrt{HYP.^2 - ALT.^2}$

$ALT. = \sqrt{HYP.^2 - BASE^2}$

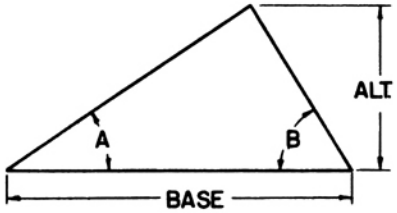
Fig. 3



BASE

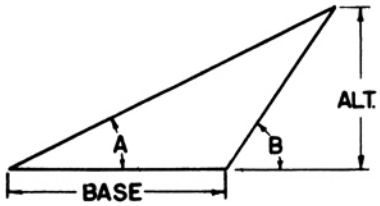
DIA. = BASE + ALT. − HYP.

Fig. 4



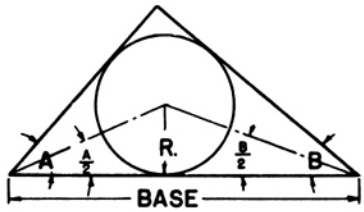
$ALT. = \frac{BASE}{COT.\ "A" + COT.\ "B"}$

Fig. 5



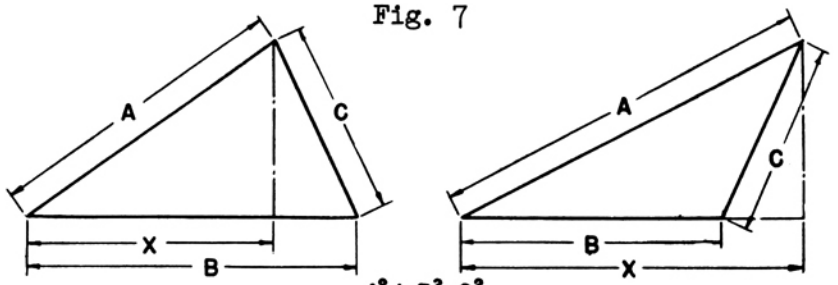
$ALT. = \frac{BASE}{COT.\ "A" - COT.\ "B"}$

Fig. 6



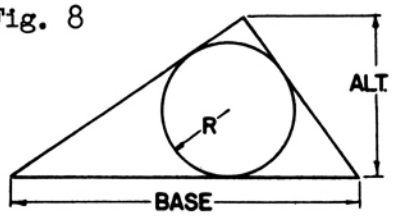
$RAD. = \frac{BASE}{COT.\frac{A}{2} + COT.\frac{B}{2}}$

Fig. 7



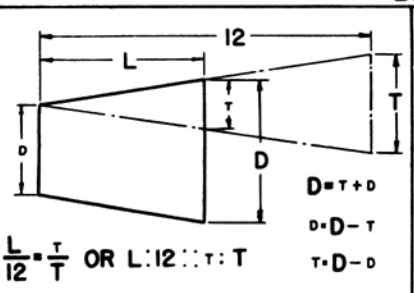
$X = \frac{A^2 + B^2 - C^2}{2B}$

Fig. 8



PERIMETER : BASE :: ALT : R.

$R = \frac{BASE \times ALT.}{PERIMETER}$



D = T + D

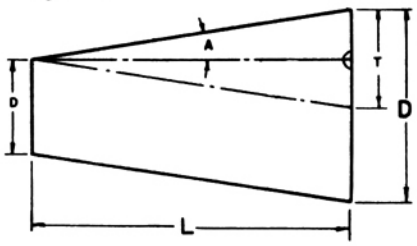
D = D − T

T = D − D

$\frac{L}{12} = \frac{T}{T}$ OR L : 12 :: T : T

Fig. 9

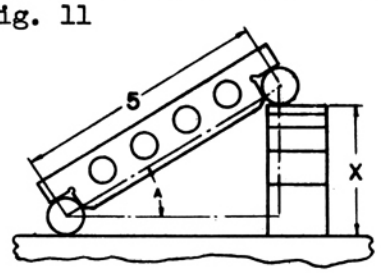
Fig. 10



T = D − D

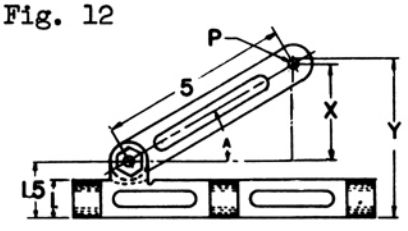
TAN. ∠ A = T ÷ 2L

Fig. 11



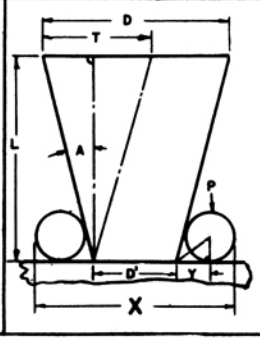
X = 5 X SINE OF ∠ A

Fig. 12



A = INCLUDED ∠ P = PLUG SIZE

X = 5 X SIN INC. ∠ Y = X + 1.5 + $\frac{P}{2}$



P = PLUG SIZE

T = TAPER PER. FT.

$\frac{T}{24} = TAN. \angle A$

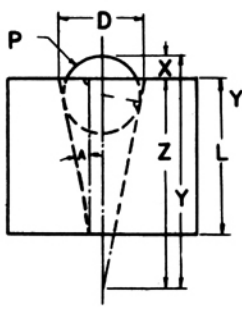
T = 2 (L X TAN ∠ A)

D' = D − T

$Y = \frac{P}{2} \times COT. \frac{90° - A}{2}$

X = D' + 2Y + P

Fig. 13



$TAN. \angle A = \frac{T}{24}$

$Y = (\frac{P}{2} CSC. \angle A) + \frac{P}{2}$

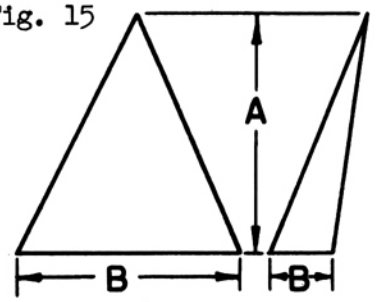
$Z = \frac{D}{2} COT. \angle A$

X = Y − Z

Fig. 14

TRIANGLE

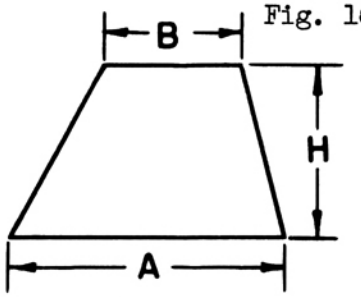
Fig. 15



$$\text{AREA} = \frac{1}{2} \text{ B X A}$$

TRAPEZOID

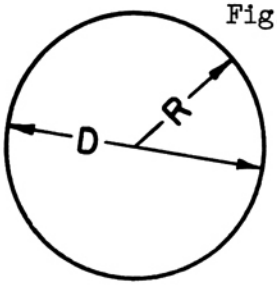
Fig. 16



$$\text{AREA} = \frac{1}{2} (A+B) H$$

CIRCLE

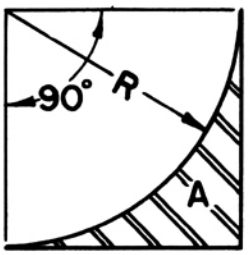
Fig. 17



$$\text{AREA} = 3.1416 R^2$$

FILLET

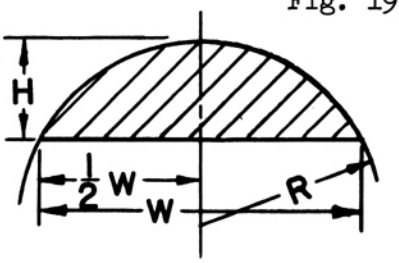
Fig. 18



$$A = R^2 - \frac{3.1416 R^2}{4}$$

SEGMENT

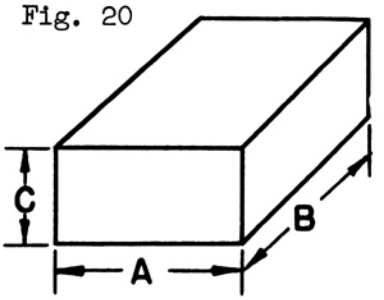
Fig. 19



$$R = \frac{(\frac{1}{2}W)^2 + H^2}{2H}$$

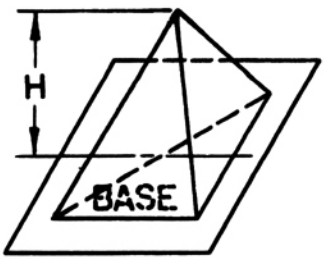
RECTANGULAR PRISM

Fig. 20



$$\text{VOLUME} = ABC$$

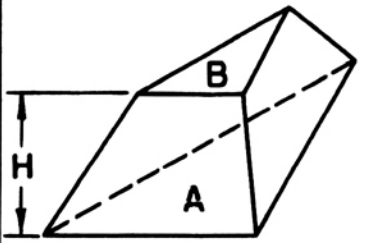
TRIANGULAR PYRAMID



$$\text{VOLUME} = \frac{\text{AREA OF BASE X H}}{3}$$

Fig. 21

FRUSTUM OF PYRAMID

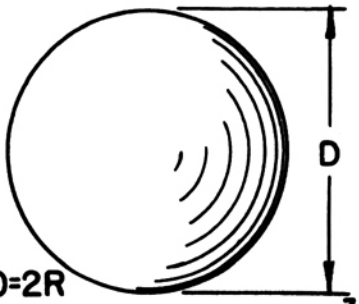


$$\text{VOLUME} = \frac{H(A+B+\sqrt{AB})}{3}$$

Fig. 22

SPHERE

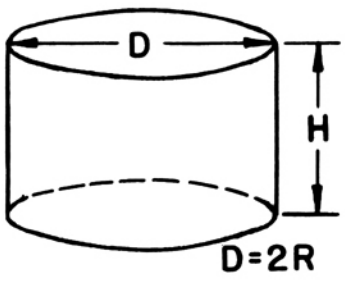
Fig. 23



$$\text{VOLUME} = \frac{4 \text{ X } 3.1416 R^3}{3}$$

CYLINDER

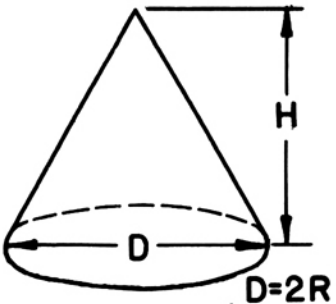
Fig. 24



$$\text{VOLUME} = 3.1416 R^2 \text{ X } H$$

CONE

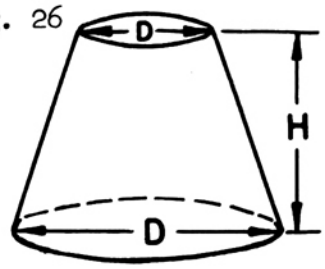
Fig. 25



$$\text{VOLUME} = \frac{3.1416 R^2 \text{ X } H}{3}$$

FRUSTUM OF CONE

Fig. 26



$$\text{VOL.} = \frac{3.1416 H(D^2+d^2+Dd)}{3}$$

Chapter 3

SMALL TOOLS

The condition in which a mechanic keeps the various tools he uses determines his efficiency as well as the judgment that others pass upon him in his daily work. A workman is always judged by the way he handles his tools. For instance, an ordinary steel rule which is battered or otherwise damaged so that it is difficult to read will often cause a workman to scrap a piece of work, thereby losing considerable time and money. A caliper with points bent out of shape will inevitably cause trouble if used in that condition. A divider with points dull or loose in the joint cannot be made to do satisfactory work. A micrometer that has been strained out of shape will not give an accurate measurement.

Every mechanic should have a tool box of his own where he keeps his tools when he is not using them. There should be a place for every tool and each tool should be kept in its place.

All tools should be wiped clean before they are placed in the tool box, and if not to be used again for some time they should be oiled to prevent rusting.

Tools that are being used on the machine or bench should be kept within easy reach of the operator and placed so that they cannot fall on the floor. They should never be placed on the finished parts of a machine.

1. Describe the most common hammers and tell the purpose for which each is used.

A. The most common hammers are the ball peen (Fig. 27), straight peen (Fig. 28), cross peen (Fig. 29), and claw hammer (Fig. 30). Hammers are made with wooden handles and steel heads and vary in size from 6 ounces to $2\frac{1}{2}$ pounds. The principal parts of a hammer are the peen, eye, face, and post. The upper part of the machinists' hammer, called the peen, is made in three common shapes--the ball peen for riveting and the straight and cross peen types for swaging. The eye is the hole that receives the handle, the face is the lower part of the head, and the post is the portion between the face and the eye. The claw hammer is used by the carpenter for driving or pulling nails.

Fig. 27

Fig. 28

Fig. 29

Fig. 30

2. What are soft hammers and why are they used?

A. Hammers with heads made of lead, copper, babbitt (Fig. 31) or rawhide (Fig. 32) are known as soft hammers. They are used to seat work in a machine vise, to drive a mandrel, or in any similar operation where the steel hammer might injure the work.

Fig. 31

Fig. 32

3. Why should a hammer handle be gripped near the end?

A. A hammer handle should be gripped near

the end so that full leverage may be obtained. A solid blow cannot be delivered when it is held too close to the head.

4. Why does the eye in a hammer head taper from each end toward the middle?

A. The hammer handle is formed to fit one end of the tapered eye and prevent the head from slipping up the handle, and in the other end of the tapered eye the handle is expanded by wedges to fit the eye and hold the head securely.

5. Why must a hammer handle be set square with the head?

A. The hammer handle must be set square with the head to insure the proper balance.

6. What is meant by "peening"?

A. Peening is the stretching of metal by hammering, such as hammering the end of a rivet or the end of a bolt so the nut will not jar loose. Peening is used to stretch babbitt to fit tightly in a bearing, to straighten bars by stretching the short side, and in numerous other operations.

Goggles must be worn when chipping, grinding, working near cyanide pots, etc. (Fig. 33). The chipping guard (Fig. 34) should be placed so as to protect fellow workmen as well as machinery.

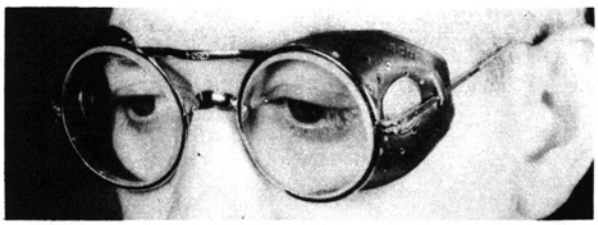


Fig. 33. Goggles

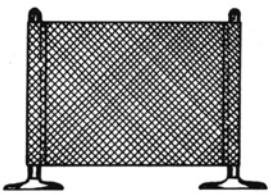


Fig. 34. Chipping Guard

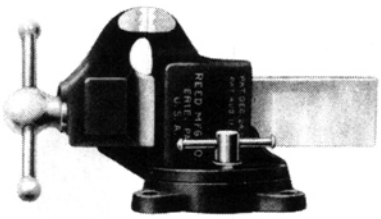


Fig. 35. Vise

A vise (Fig. 35) is used for holding work while it is being finished, assembled, etc., at the bench. Do not hammer a vise handle. When holding work in a vise, put a packing block of wood or metal under it to keep it from working down in the vise. Protect finished work with brass or copper jaw caps (Fig. 36). Place the work so that it can be chiseled toward the solid jaw. Refer to Chisels and Chipping, page 47, for an illustration of the proper use of these tools.

Fig. 36. Vise Jaw Cap

7. Describe the hack saw.

A. The hack saw (Fig. 37) consists of a frame and a thin steel blade with teeth formed on one edge. It is six to twelve inches long, approximately one-half inch wide, and usually about .027" thick.

Fig. 37. Hack Saw

8. What is a chisel?

A. A chisel is a tool or instrument made from octagon-shaped steel, having a cutting edge at one end of the blade, and used in dressing, shaping, or cutting. It is driven by a hammer or mallet. Some of the different types are shown below and on the following page.

Fig. 38. Flat Cold Chisel

The flat cold chisel is used for chipping flat surfaces, and often for cutting thin sheet metal.

The cape chisel (Fig. 39) is used for chipping grooves, keyways, and often for slots where a sturdy narrow chisel is needed.

Fig. 39. Cape Chisel

The round nose chisel is used for roughing out small concave surfaces of filleted corners, etc.

Fig. 40. Round Nose Chisel

The oil groove chisel is used for chiseling oil grooves, etc.

Fig. 41. Oil Groove Chisel

The diamond point chisel is used for cutting V-shaped grooves or for chipping in corners.

Fig. 42. Diamond Point Chisel

The star drill is really a multiple-pointed chisel and is used for drilling stone and concrete.

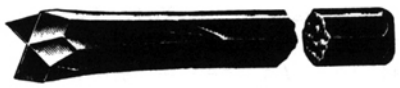


Fig. 43. Star Drill

The drift punch is used for aligning holes for bolts or rivets.

Fig. 44. Drift Punch

The pin punch is used to drive taper pins, cotter pins, etc.

Fig. 45. Pin Punch

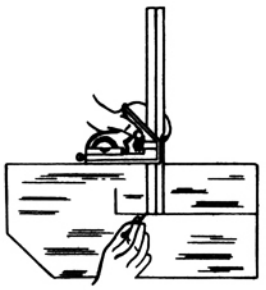
The prick punch is used to mark scribed or layout lines with small indentations.

Fig. 46. Prick Punch

The center punch is used to mark the location for the drill and assist in starting it properly.

Fig. 47. Center Punch

Fig. 48. Scriber



The scriber is used to mark lines in measuring or on layout work.

Fig. 49

9. What is a wrench?
   A. A tool with jaws or openings for turning or twisting bolts, nuts, etc.

10. How are wrenches named?
   A. (1) From their shape, as "S" wrench, angle wrench, etc.
   (2) From the object on which they are used, as tap wrench, pipe wrench, etc.
   (3) From their construction, as spanner wrench, ratchet wrench, etc.
   Some of the most common wrenches are shown on the following page.

11. Explain the use of each of the following wrenches: single-end, double-end, closed-end, adjustable hook spanner, adjustable pin face, T-socket, off-set socket, ratchet, pipe, strap, chuck key, hollow set-screw, and monkey wrench.

The single-end wrench shown in Fig. 50 is used on jobs requiring a wrench with one size only.

Fig. 50. Single-end Wrench

The double-end wrench shown in Fig. 51 is used on jobs that require a wrench which has openings of different sizes.

Fig. 51. Double-end Wrench

The use of the closed-end wrench shown in Fig. 52 is much the same as that of the single-end wrench. The closed end of the wrench eliminates the danger of the jaws spreading.

Fig. 52. Closed-end Wrench

The adjustable hook spanner wrench (Fig. 53) is used on nuts having notches cut in the periphery to receive the hook located at the end of the wrench.

Fig. 53. Adjustable Hook Spanner Wrench

The adjustable pin face wrench is used to adjust nuts having holes in their face to accommodate pins in the end of the adjustable legs (see Fig. 54).

Fig. 54. Adjustable Pin Face Wrench

The T-socket wrench (Fig. 55) is made with different types of sockets, such as square, hexagon, and octagon. It is generally used on jobs where the nuts are almost inaccessible, as on engines, motors, bodies, wire wheels, etc.

Fig. 55. T-Socket Wrench

The off-set socket wrench (Fig. 56) is used on nuts requiring a greater leverage than that which can be obtained with the T-socket wrench, or on jobs where the T-socket wrench cannot be used.

Fig. 56. Off-set Socket Wrench

The ratchet wrench (Fig. 57) is similar to the socket wrench except that the handle works as a ratchet. This is especially useful when only a short swing of the handle is permissible. Another advantage of this wrench is that it is not necessary to remove it until the bolt or nut is tight.

Fig. 57. Ratchet Wrench

The Stillson pipe wrench (Fig. 58) and the chain pipe tongs (Fig. 59) are used to turn pipes. Be careful in using these wrenches that the finish on plated pipes is not scratched or otherwise damaged.

Fig. 58. Stillson Pipe Wrench

Fig. 59. Chain Pipe Tongs

The strap wrench (Fig. 60) is used for turning plated pipes, removing bezels, etc., or for revolving any job on which the surface finish must be preserved.

Fig. 60. Strap Wrench

The chuck key (Fig. 61) is used for adjusting chuck jaws.

Fig. 61. Chuck Key

The hollow set-screw wrench (Fig. 62) is used to adjust hollow (safety) set-screws.

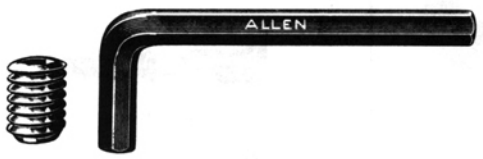


Fig. 62. Hollow Set-Screw Key

The monkey wrench is a heavy adjustable wrench for use on heavy work (see Fig. 63). Its use on small tools is not encouraged. The three following rules for the use of the monkey wrench are very important:

1. Never hammer with the solid jaw of the wrench.

2. Always adjust the movable jaw so that it is tight against a nut.

3. Point the jaws in the direction of the force applied. This will prevent the jaws from springing and the wrench will be less likely to slip off a nut.

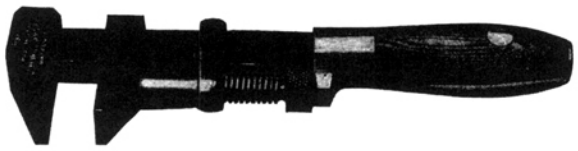


Fig. 63. Monkey Wrench

Do not use a pair of pliers instead of a wrench (see Fig. 64).

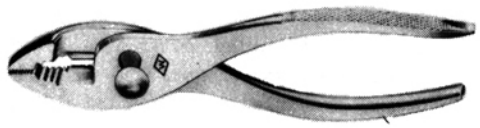


Fig. 64. Pliers

12. What is a screw driver?
A. A screw driver is a metal blade with a handle on one end with the other end flattened to fit screw slots.

13. What kind of steel is generally used to make screw drivers?
A. Tool steel is generally used. Screw drivers under one-half inch in size are usually made of round stock (see Fig. 65).

Fig. 65

14. Why are the larger sizes of screw drivers made with square blades?
A. The blade of the screw driver shown in Fig. 66 is made square so that a wrench may be applied to assist in turning a screw.

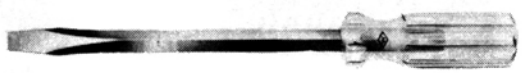


Fig. 66

15. What is the purpose of the double-end off-set screw driver shown in Fig. 67?

A. This type of screw driver is used for turning screws that cannot be reached with a straight screw driver.

Fig. 67

16. How should the blade of any screw driver be ground?

A. A screw driver blade should be ground so that the faces will be almost parallel with the sides of the screw slot. Fig. 68 shows a screw driver correctly ground.

If a blade is incorrectly ground as shown in Fig. 69, it has a tendency to slip out of the slot and leave a burr.

Excessive heat due to grinding, shown by a blue color, will draw the temper and cause the driver to become soft.

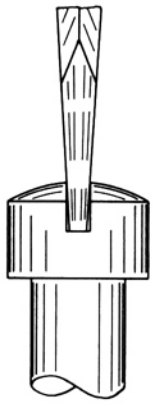


Fig. 68

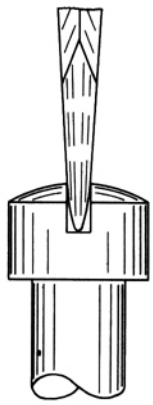


Fig. 69

17. Describe the helical ratchet screw driver shown in Fig. 70.

A. The helical ratchet screw driver drives or draws screws by pushing on the handle. It also has a ratchet movement and can be locked rigid. It may be changed from right-hand to left-hand or locked by moving the shifter.

Fig. 70

18. What are the three most important classes of screws and what are the names of the principal types of screw heads in each class?

A. Machine screws, set screws, and wood screws are the most important classes. Fig. 71 names and illustrates the principal screw heads in these classes.

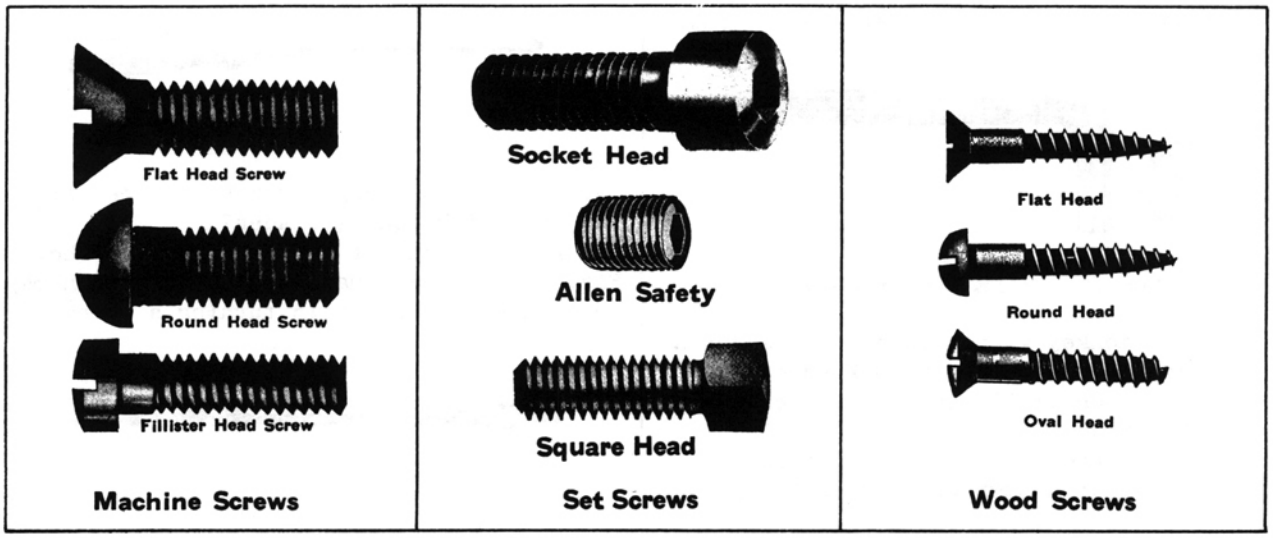


Fig. 71. Different Types of Screws and Heads

19. What do flat head screws and fillister or socket head screws have in common?

A. These screws have heads that must be recessed into the work. The 82° countersink is used to form the recess for a flat head screw. A counterbore, the diameter of which is slightly larger than the screw head, and a pilot, the diameter of which is equal to the screw body size, is used to form the recess for a socket head screw.

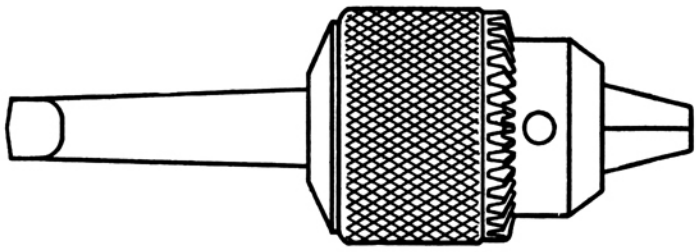


Fig. 76. Jacobs Drill Chuck

20. What is a drill? Describe some of the various types of drills and drill equipment.

A. A drill is a round steel shaft with grooves formed on its periphery, one end, known as the point, is ground to form suitable cutting edges (see Figs. 72 and 73). The shank end of a drill may be either tapered or straight. Most drills have a tapered shank, which can be fitted into a spindle, socket, or sleeve (see Figs. 74 and 75). All drills with a diameter less than 1/8 have straight shanks, however, straight shanks in other sizes are available. Drills with straight shanks are held in a drill chuck (Figs. 76 and 77).

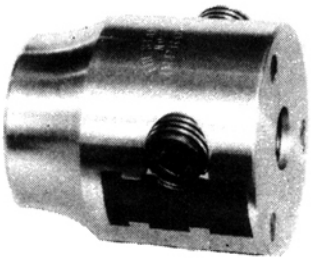


Fig. 77. Skinner Drill Chuck

Figs. 78 and 79 show two drills for drilling wood, steel, or other material, one driven by hand and the other by an electric

Fig. 72. Taper Shank Drill

Fig. 73. Straight Shank Drill

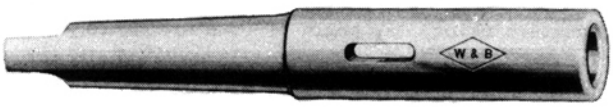


Fig. 74. Socket

Fig. 75. Sleeve

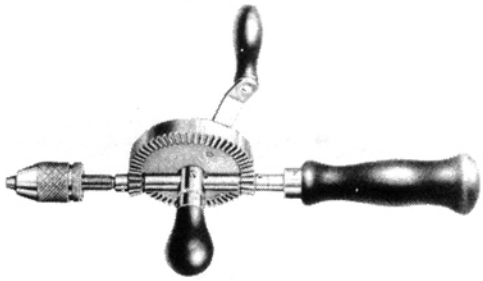


Fig. 78. Hand Drill

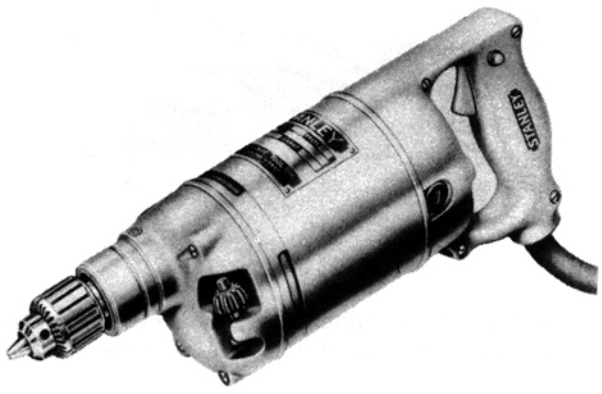


Fig. 79. Portable Electric Drill

motor. The drill holder in Fig. 80 is used to hold a drill while drilling in a lathe. A drill drift is used to remove drills from sleeves and sockets. A plain drift is shown in Fig. 81 and a safety drift in Fig. 82. The drill vise in Fig. 83 is used to hold work while it is being drilled.

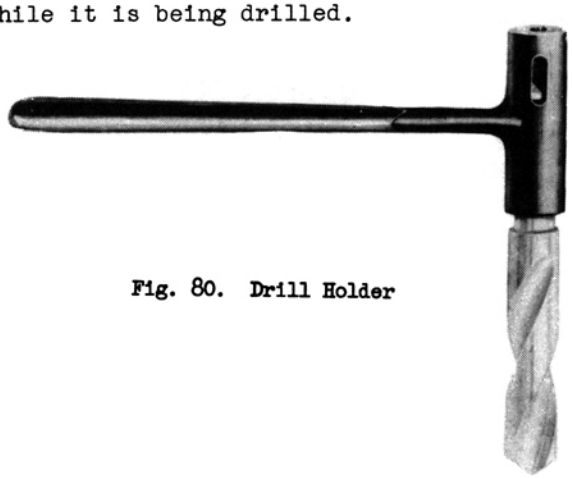


Fig. 80. Drill Holder

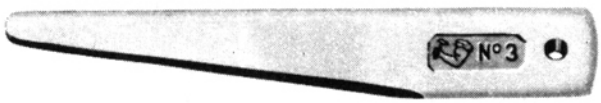


Fig. 81. Plain Drill Drift

Fig. 82. Safety Drill Drift

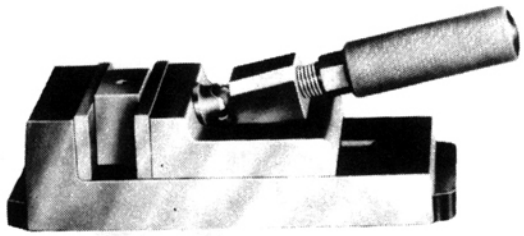


Fig. 83. Drill Vise

21. How are the countersink, the counterbore, and the combined drill and countersink used?

A. The countersink, shown in Fig. 84, is used for countersinking the tops of holes which are to receive flathead screws or bolts.

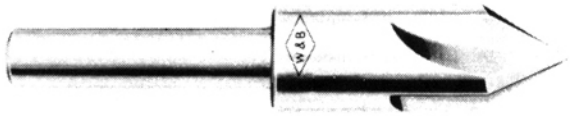


Fig. 84. Countersink

The counterbore shown in Fig. 85 is used for enlarging a hole on the axis of one already drilled, so that the head of the bolt or cap screw will come flush with the surface of the work.

Fig. 85

Fig. 86 shows a combined drill and countersink which is used for drilling the center holes in work which is to be mounted on centers. It is also used to spot the center in a piece that is to be drilled.

Fig. 86

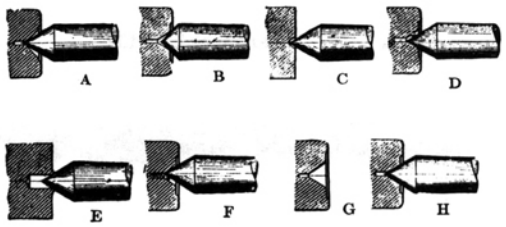


Fig. 87. Examples of combined drill and countersinking. A, angle of countersink too large; B, angle of center too large; C, hole not large enough; D, hole upset; E, hole not countersunk; F, chips in center hole; G, correct center hole as shown by fit at H.

22. Explain what a reamer is used for and name 3 types of reamers.

A. A reamer is a tool for enlarging a hole, round, straight, smooth, and up to size. Fig. 88 shows a hand reamer which is a finishing tool used to remove from .002" to .003" stock. The machine reamer shown in Fig. 89 is used for reaming in the drill press

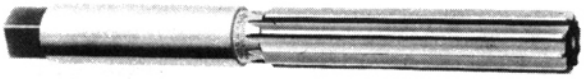


Fig. 88. Hand Reamer

Fig. 89. Machine Reamer

or lathe. It removes approximately 1/64" stock. Fig. 90 shows an expansion reamer. This reamer is adjustable for reaming holes of special size. Use a solid reamer whenever possible.

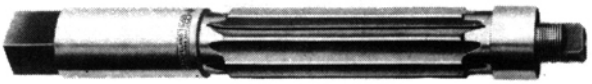
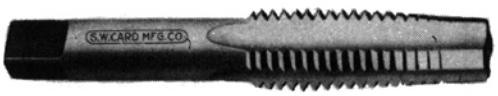


Fig. 90. Expansion Reamer

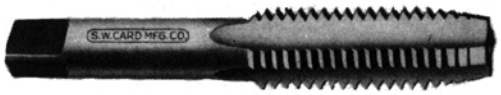
Chart for the expansion of reamers

Size	Limit	Size	Limit
$\frac{1}{4}$" to 15/32"	.005"	1" to 1 23/32"	.010"
$\frac{1}{2}$" to 31/32"	.008"	1 3/4" to 2$\frac{1}{2}$"	.012"

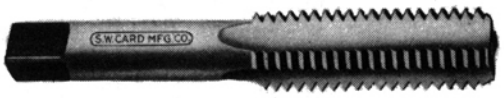
23. Name some of the common types of taps and the tools for revolving them.



(a)



(b)



(c)

Fig. 91. Hand taps for internal threading: (a) taper; (b) plug; (c) bottoming.

A T tap wrench used for revolving small taps is shown in Fig. 92. Fig. 93 shows a hand tap wrench for revolving larger taps. Fig. 94 shows a tapping chuck which is used for holding taps in machine tapping operations.

24. How can a broken tap be removed?

A. The tap extractor shown in Fig. 97 is used to remove a broken tap. It must be used carefully, as it is a fragile tool.

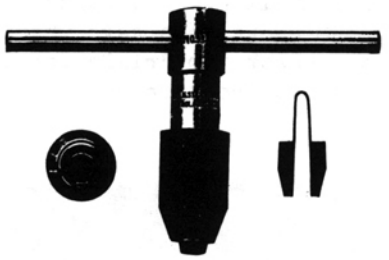


Fig. 92. T Tap Wrench

Fig. 93. Hand Tap Wrench

Fig. 94. Tapping Chuck

Fig. 95. Nut or Extension Tap

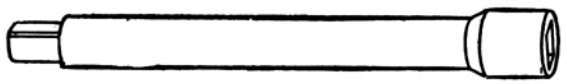


Fig. 96. Tap Extension

Fig. 97. Tap Extractor

25. What is a drill and tap gage?

A. Fig. 98 shows this gage which is used to select and check tap drills for the most commonly used machine screws.

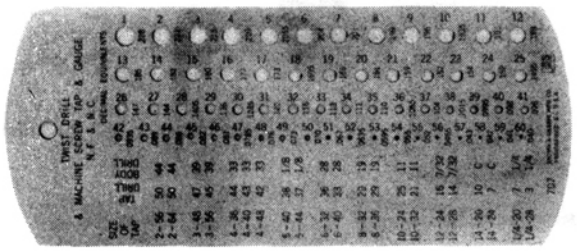


Fig. 98. Drill and Tap Gage

26. What tools are used in cutting an external thread by hand?

A. External threading is done with a die, such as the one shown in Fig. 99. This die is inserted in the die stock of Fig. 100 and turned by hand.

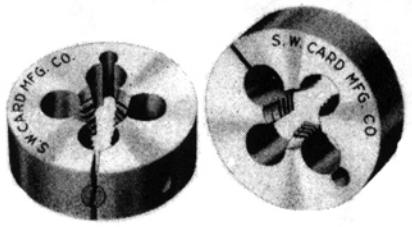


Fig. 99. Round Split Die

Fig. 100. Die Stock

27. What is an "ezy-out"?

A. The ezy-out illustrated in Fig. 101 is used to remove broken bolts, screws, etc., from holes. In using it, a hole is first drilled in a broken bolt or screw, the size being a little smaller than the minor diameter of the thread. An ezy-out of the proper size is then inserted and revolved counter-clockwise (see Fig. 103).

Fig. 101. Ezy-out

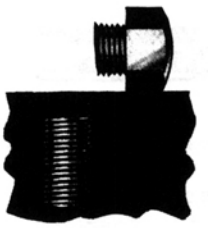


Fig. 102.
Broken Screw

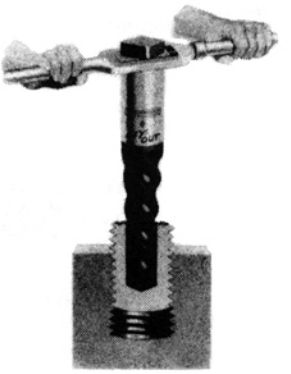


Fig. 103.
Using the Ezy-out

28. What are the different types of rules generally used in the shop?

A. Figs. 104 to 107 illustrate several rules which are commonly used in the shop.

Fig. 104. Six Inch Steel Rule

Fig. 105. Hook Rule

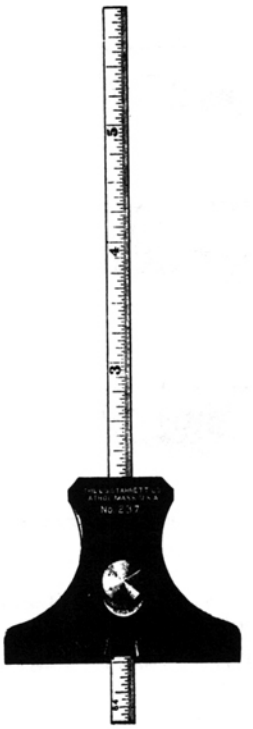


Fig. 106.
Rule Depth Gage

Fig. 104 shows the common six inch steel rule, which is used in many ways. Fig. 105 shows the hook rule, used for measuring inside dimensions where the hook is an advantage. The rule depth gage of Fig. 106 is used to find the depth of holes, slots, etc. The tape-rule in Fig. 107 is used where a flexible measuring instrument is desired. The boxwood folding rule, the zig-zag rule, and the carpenters' steel square, which are commonly used by carpenters, are illustrated on pages 27 and 28.

29. Describe the combination square and some of the equipment used with it.

A. Fig. 108 shows the

Fig. 107. Steel Tape-Rule

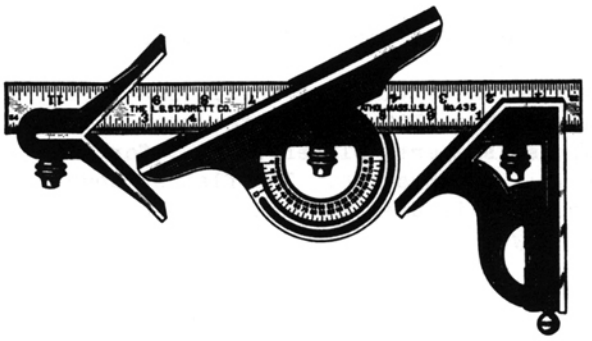


Fig. 108. Combination Set

combination set, consisting of the combination square, a center head, and a protractor (the center head and the protractor being frequently used with the combination square). This set has a multitude of uses, especially in laying out centers and angles. Fig. 109 shows several common applications of the combination square and one application of the center head.

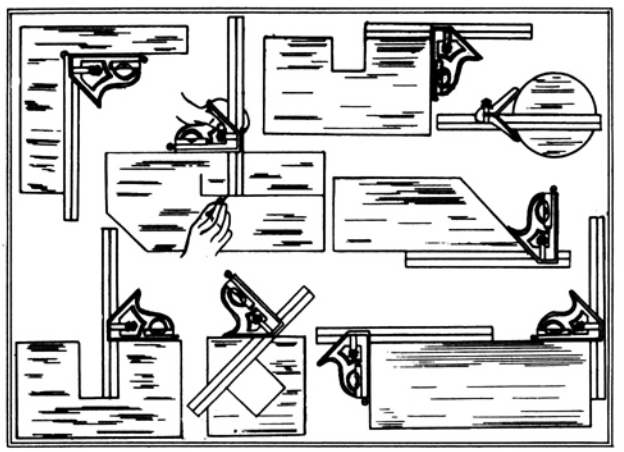


Fig. 109. Combination Square and Center Head Applications.

30. What devices are commonly used to hold work?

A. In addition to the drill vise in Fig. 83, several types of clamps are used to hold work. Fig. 110 shows a small toolmakers' steel clamp, which has two interchangeable blocks for varying the capacity of the clamp (the small block being shown on the left). Fig. 111 shows toolmakers' parallel clamps and Fig. 112 shows a C clamp. The V blocks and clamps shown in Fig. 113 are used frequently in holding round work.

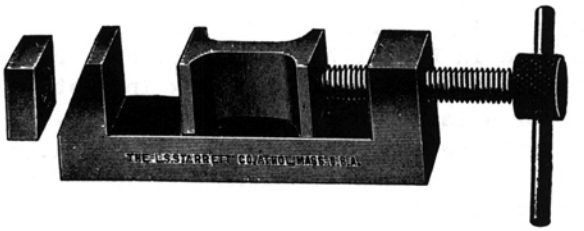


Fig. 110. Toolmakers' Steel Clamps

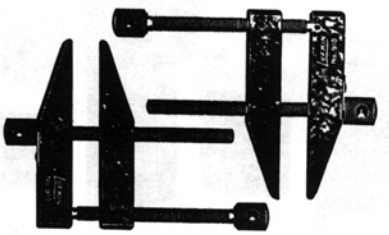


Fig. 111. Toolmakers' Parallel Clamps

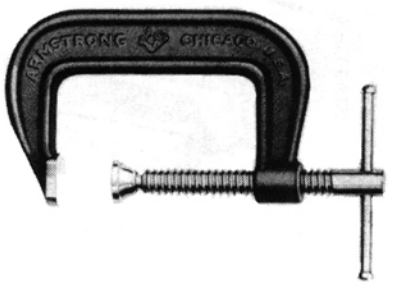


Fig. 112. C Clamp

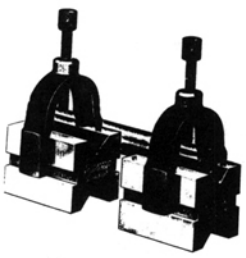


Fig. 113
V Blocks and Clamps

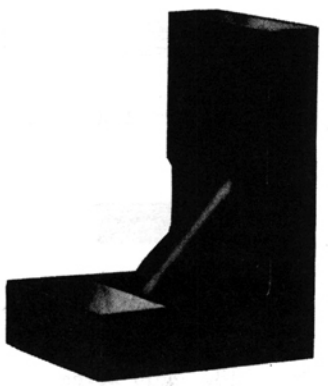


Fig. 114. Angle Plate

Several other devices, besides clamps, are used for holding work in various ways. The angle plate in Fig. 114 is used to hold work for laying out or machining. The box

parallel or cube shown in Fig. 115 is used mostly in the grinding department for holding work while 90° angles are being ground, and the parallels in Fig. 116 are used to support work in parallel planes for laying out, shaping, milling, grinding, checking, etc. The cat head in Fig. 117 is used for supporting irregular shaped revolving work in a steady rest of a lathe, the work being adjusted centrally by set screws. The hold downs illustrated in Fig. 118 are generally used for holding work in a shaper vise. The step block in Fig. 119 is used for clamping work, as shown in Fig. 120.

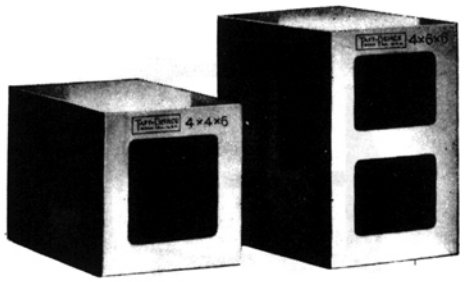


Fig. 115. Box Parallel or Cube

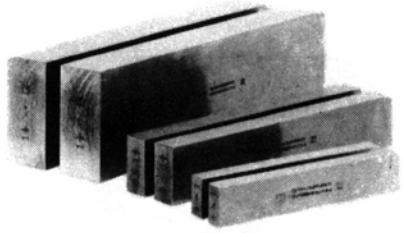


Fig. 116. Parallels

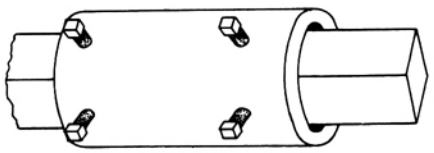


Fig. 117. Cat Head

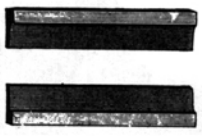


Fig. 118. Hold Downs

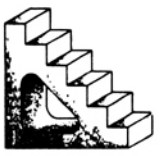


Fig. 119. Step Block

31. Explain how a surface plate and a straight edge are used.
A. The surface plate shown in Fig. 121 is used to check flat surfaces. The straight edge in Fig. 122 is used to check the ways of lathes, shapers, planers, etc., for being flat as well as straight. The flat scraper (see Fig. 158) is used to remove the high spots of the ways.

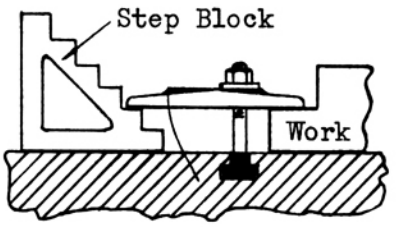


Fig. 120. Application of Step Block

Fig. 121. Surface Plate

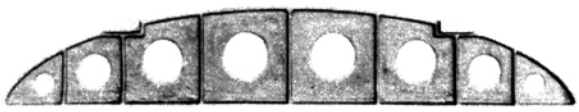


Fig. 122. Straight Edge

When the surface plate or straight edge is being used, it is coated lightly with a marking material such as Venetian red or Prussian blue, and then rubbed over the surface that is to be trued. The marking material is left on the high places, thus showing the high spots and are to be removed with a scraper. This operation is repeated until the surface shows a good bearing at all points. Small work is rubbed on the plate. Every part of the surface plate should be used as evenly as possible, for rubbing in one place will wear the surface unevenly.

32. Name several tools and pieces of equipment commonly used on lathe work.
A. Fig. 123 shows the straight tool holder, used for threading and general turning. Figs. 124 and 125 show left-hand and right-hand tool

Fig. 123. Straight Tool Holder

Fig. 124. Left-hand Tool Holder

Fig. 125. Right-hand Tool Holder

Fig. 126. Straight Cutting-off Tool Holder

Fig. 127. Right-hand Off-set Cutting-off Tool Holder

Fig. 128. Threading Tool Holder with Spring Head

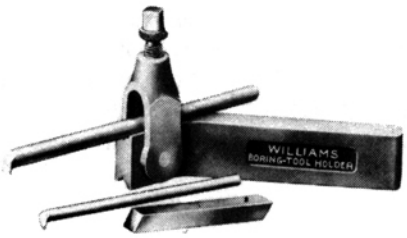


Fig. 129. Forged Boring Tool Holder

holders. The left-hand tool holder is used for turning material with a large diameter and for facing toward dead center (do not take a heavy cut toward dead center). The right-hand tool holder is used for facing toward live center and for turning close to the chuck.

Figs. 126 and 127 show holders used for cutting-off work. The holder in Fig. 128 is used for producing threads of a fine finish with close limits.

Fig. 129 shows a holder for a forged boring tool. The knurling tool in Fig. 130 is used to produce a knurled finish, as shown in Fig. 131.

The boring bar holder in Fig. 132 is used to hold large heavy duty boring bars. The lathe center (Fig. 133) is used to support revolving work and the lathe mandrel (Fig. 134) is used to support work with standard holes while it is being machined.

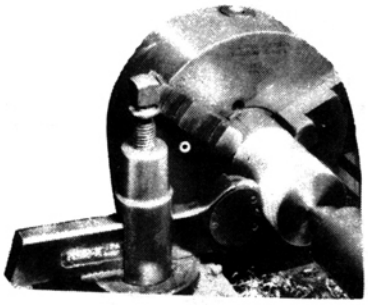


Fig. 131. Knurling Tool in Use

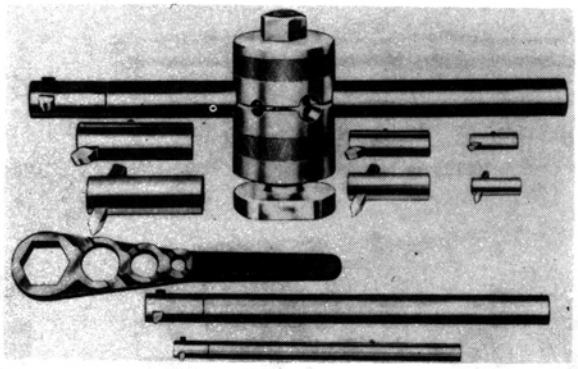


Fig. 132. 3-Bar Boring Tool Set

Fig. 130. Knurling Tool

Fig. 133. Lathe Center

Fig. 134. Lathe Mandrel

The expanding mandrel in Fig. 135 is used to support work with special size holes while machining it. The bent tail lathe dog (Fig. 136) is used for driving round or regular work (square, hexagon, etc.) and the clamp dog (Fig. 137) for rectangular work.

33. Name some of the equipment and cutters used on the milling machine.

A. The milling machine arbor in Fig. 138 is used for revolving and driving milling machine cutters. The micrometer offset boring head shown in Fig. 139 is adjustable and is used to drive a boring tool. The Woodruff keyway cutter and holder in Fig. 140 are used in cutting Woodruff keyways.

The slitting saw in Fig. 141 is used for milling narrow slots, the plain cutter for milling slots the same width as the cutter, the side cutter for milling vertical and horizontal surfaces, and the angular cutter for milling angles. The convex cutter is used for cutting concave surfaces and the concave cutter for cutting convex surfaces. The involute form cutter is used for cutting gear teeth. The inserted tooth side cutter is the most economical of the large cutters, as broken teeth can be replaced easily. The slab cutter is used for milling flat surfaces where a large cutting area is desired. The gear hob is used on gear hobbers for cutting teeth in gears. The shell end mill is used

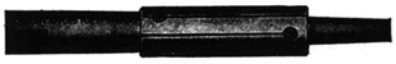


Fig. 135. Expanding Mandrel

Fig. 136. Bent Tail Lathe Dog

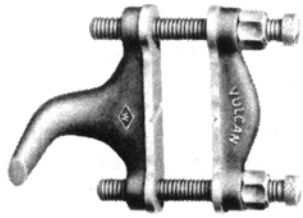


Fig. 137. Clamp Dog

Fig. 138
Milling Machine Arbor

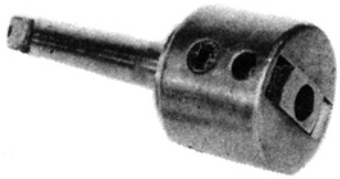


Fig. 139
Micrometer Offset Boring Head

Fig. 140
Woodruff Keyway Cutter and Holder

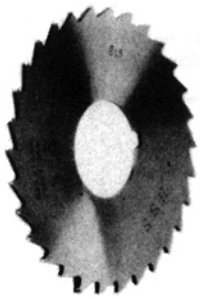


Fig. 141
Slitting Saw

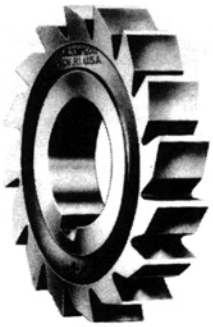


Fig. 142
Plain Cutter

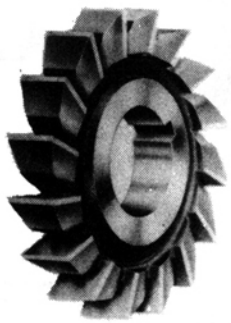


Fig. 143
Side Cutter

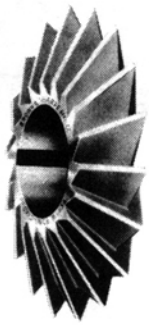


Fig. 144
Angular Cutter

Fig. 145
Convex Cutter

in end milling when a large cutting area is wanted.

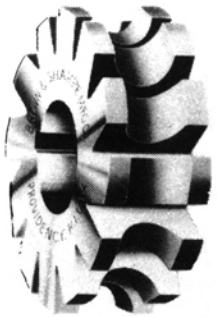


Fig. 146
Concave Cutter

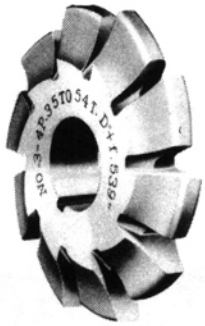


Fig. 147
Involute Form Cutter

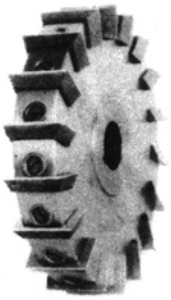


Fig. 148
Inserted Tooth Side Cutter

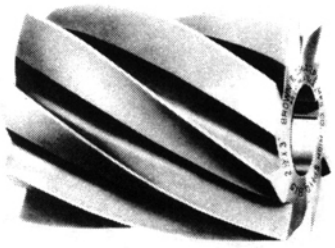


Fig. 149. Slab Cutter

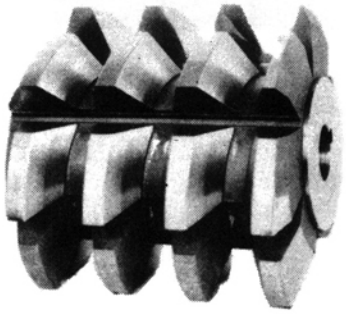


Fig. 150. Gear Hob

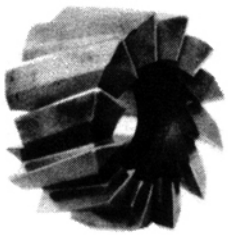


Fig. 151
Shell End Mill

The two lip end mill in Fig. 152 is used for milling slots in solid metal and the helical end mill in Fig. 153 is used for milling slots, fillets, keyways, etc.

34. What equipment is used for dressing and truing grinding wheels?

A. The metallic abrasive wheel dresser (Fig. 154) and the magazine wheel dresser (Fig. 155) are used for dressing grinding wheels. The diamond nib in Fig. 156 is used for both dressing and truing.

Fig. 152. Two Lip End Mill

Fig. 153. Helical End Mill

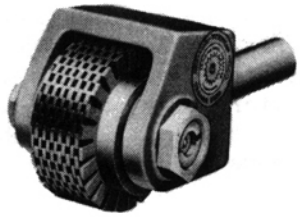


Fig. 154. Metallic Abrasive Wheel Dresser

Fig. 155. Magazine Wheel Dresser

Fig. 156
Diamond Nib

35. Name the various types of bearing scrapers.

A. The bearing scraper shown in Fig. 157 is used, as its name implies, for scraping bearings. Fig. 157A, the three-corner scraper is used mostly for removing the burrs or sharp internal edges from soft bushings, etc. Do not use a file for this operation. Flat scrapers are used to remove high spots from flat bearing surfaces.

Fig. 157. Bearing Scraper

Fig. 157A. Three-Corner Scraper

Fig. 158. Flat Scraper

The oil stone shown in Fig. 159 is used to sharpen scrapers, cutting tools, etc. Keep the surface of a stone well oiled.

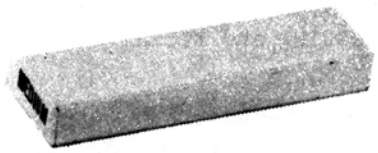


Fig. 159. Oil Stone

36. What gages, measuring instruments, and equipment are commonly used in measuring and setting up precision work in the toolroom?

A. Fig. 160 shows a micrometer, the most commonly used precision tool in the toolroom. This tool was invented in 1848 by Jean Palmer, a Frenchman, and has been improved from time to time until the modern micrometer has resulted. There are other types of micrometers besides the outside micrometer shown here, which is the type generally used. Fig. 199 shows the thread micrometer (which is an outside micrometer) and Fig. 170 shows the depth micrometer. Still another type is the inside micrometer. All of these are to be studied later in the course.

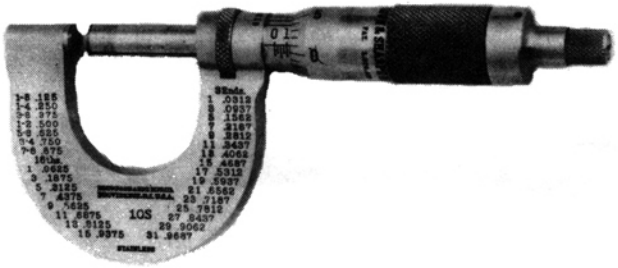


Fig. 160. Common Outside Micrometer

Fig. 161 shows a sine bar, which is used for checking angles, tapers, etc., and Fig. 162 shows an individual set of Johansson gage blocks, used for precision measuring. These blocks are rectangular pieces of tool

Fig. 161. Sine Bar

steel, hardened, ground, and finished to an accuracy within a few millionths of an inch. A full set consists of eighty-one blocks, and these will make 120,000 different size gages, in steps of .0001", from a minimum size of .200" to a maximum size of 12.000". Fig. 163 shows an application of a sine bar, using Johansson gage blocks.

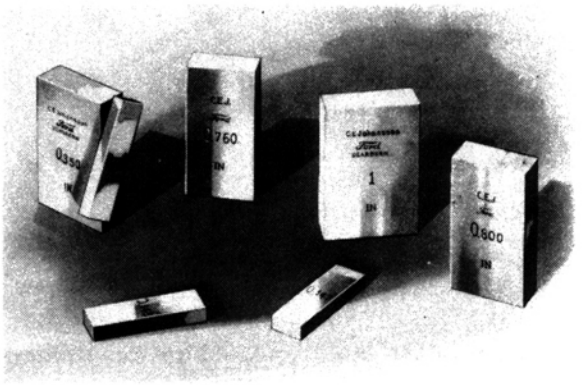


Fig. 162. Johansson Gage Blocks

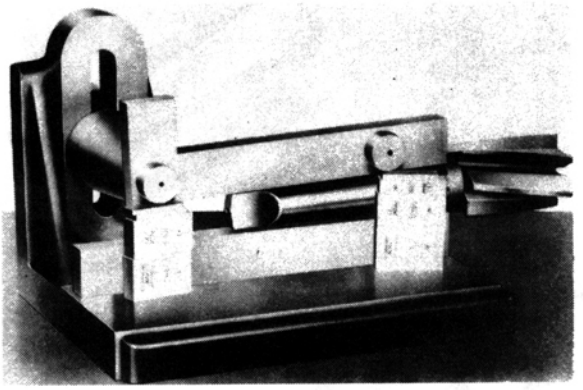


Fig. 163. Application of a Sine Bar, Using Johansson Gage Blocks.

The surface gage shown in Fig. 164 is used for layout work, for leveling and lining

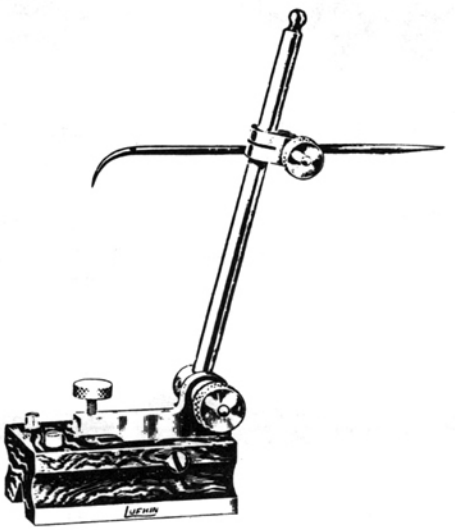


Fig. 164. Surface Gage

up work, and for locating centers on rough work. The surface gage may be set to a combination square in the following way, using the straight end of the scriber (see Fig. 165). First, set the square on the layout table, as shown in the illustration, being

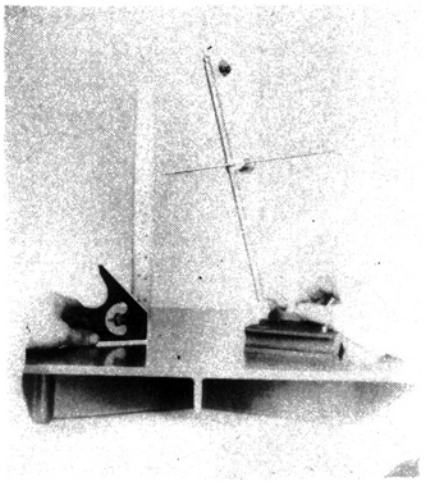


Fig. 165. Setting a Surface Gage

sure that the blade of the square is resting on the table and is clamped in place in the head. Then set the standard of the surface gage, clamp in place, adjust the scriber to the approximate height desired, and clamp in place on the standard. Finally, set the scriber against the desired index mark on the blade of the square, and, by means of the adjusting screw on top of the base of the gage, set the scriber to the desired location on the blade of the square.

Two of the many applications of the surface gage are shown in Fig. 166.

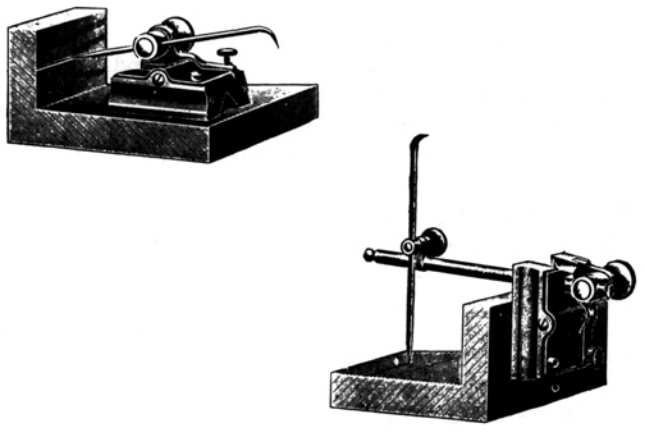


Fig. 166
Surface Gage Applications

Fig. 167 shows a planer gage, which is used for setting the cutting tool on a planer (it may also be used on bench work). This gage can be set to a micrometer (Fig. 168), surface gage, or caliper.

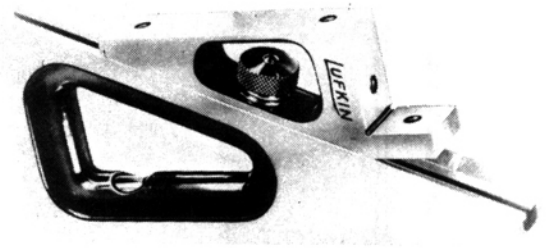


Fig. 167. Planer Gage

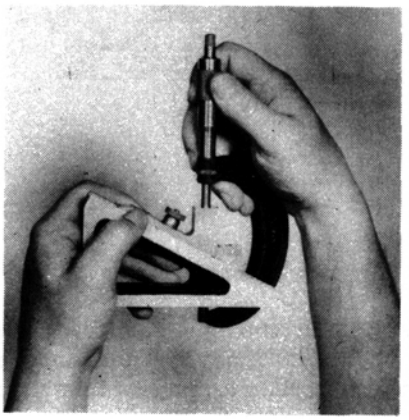


Fig. 168. Setting a Planer Gage

Figure 169 shows a solid square, which is used for checking square work.

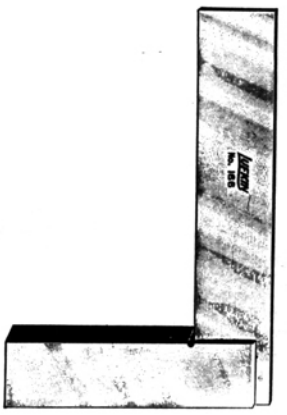


Fig. 169. Solid Square

Figs. 170 and 171 show depth gages, one with a micrometer scale and one with a vernier scale. These are used for measuring the depth of holes and grooves.

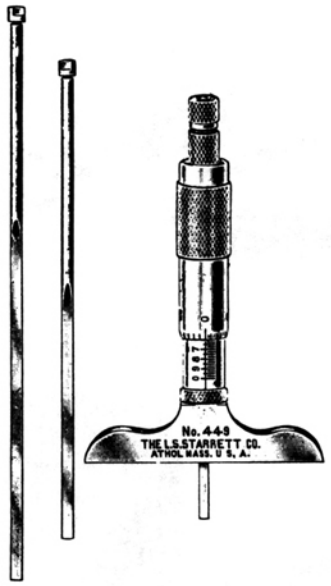


Fig. 170. Micrometer Depth Gage

Fig. 172 shows a vernier height gage, which is used for locating center distances, finding the height of projections, etc.

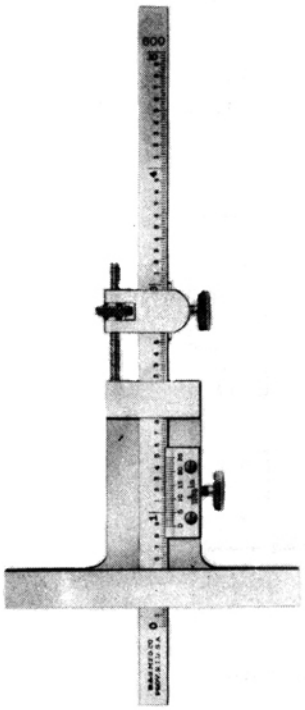


Fig. 171
Vernier Depth Gage

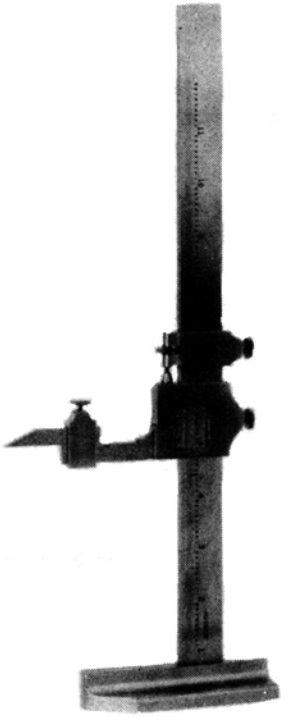


Fig. 172
Vernier Height Gage

Fig. 173 shows a vernier caliper, used to determine inside and outside dimensions to the thousandth of an inch. The caliper is graduated to read on one side for outside measurements, and on the other side for inside measurements.

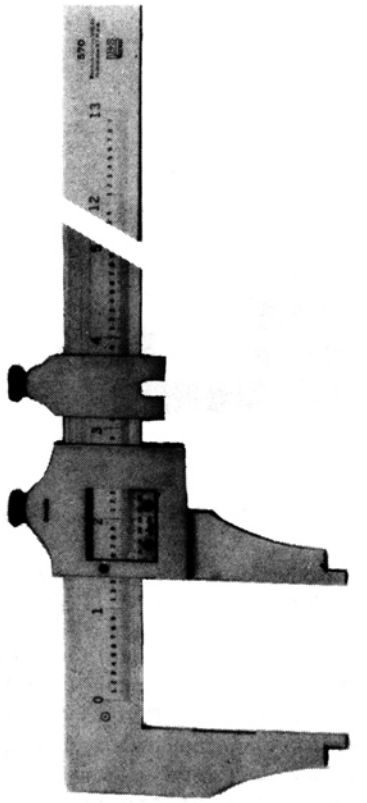


Fig. 173. Vernier Caliper

The gear tooth vernier caliper in Fig. 174 is used for finding the chordal thickness and the corrected addendum of gear teeth, worms, etc.

The principle of the Vernier, with which these last four tools are equipped, will be explained later.

Fig. 175 shows the Test Indicator, which is generally used in connection with a height gage. The indicator is mounted on a hardened steel tool post holder (only a portion of the holder being shown). The scale is graduated in thousandths, and an indicator with either a single or double scale can be obtained (the double scale can of course be read from both front and rear). The contact point is spherical in shape and can be turned 90° on either side of the indicator.

Fig. 176 shows a dial indicator, which is commonly used for general shop work. The dial alone is shown in Fig. 177. It is graduated in half-thousandths from 0 to 25 to 0, so that plus or minus measurements are indicated, and readings of less than a half-

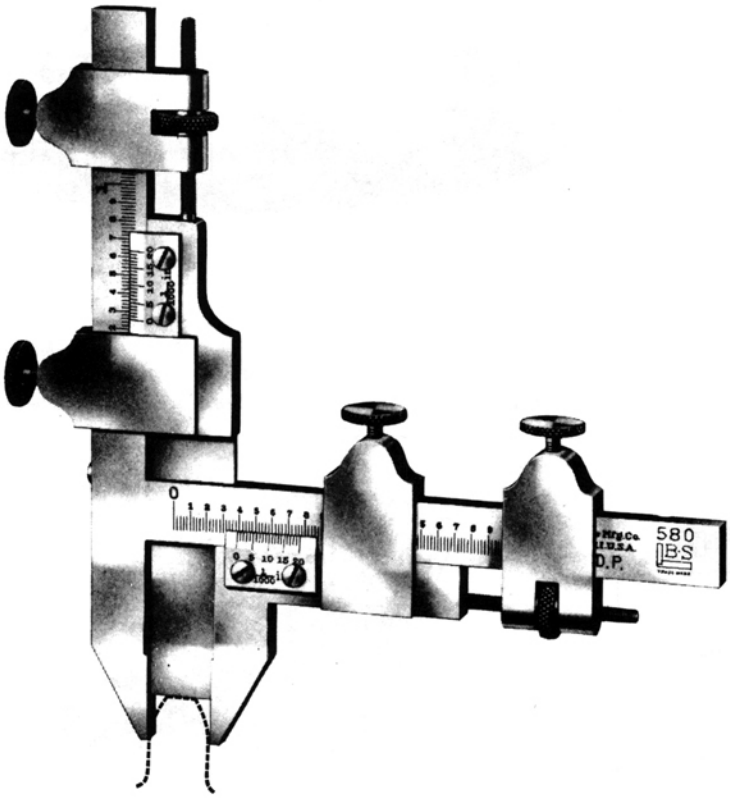


Fig. 174. Gear Tooth Vernier Caliper

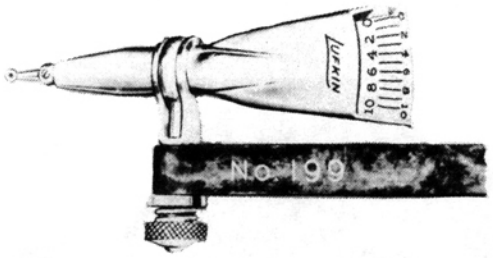


Fig. 175. Test Indicator

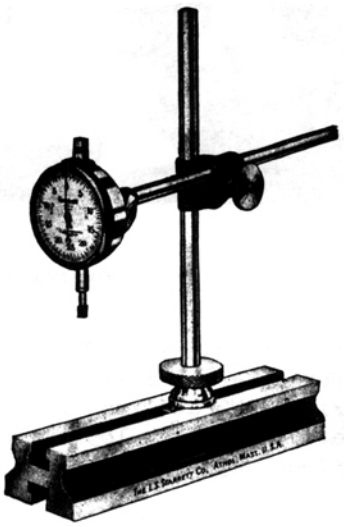


Fig. 176. Dial Indicator

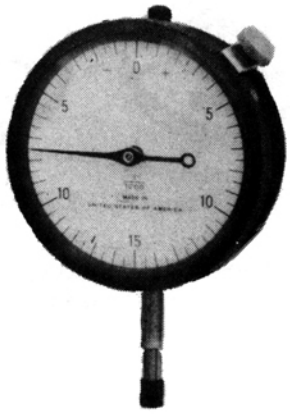


Fig. 177. Dial

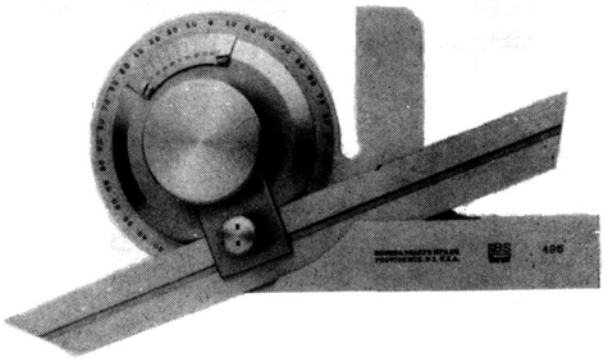


Fig. 178. Vernier Bevel Protractor

thousandth can be estimated. The dial can be turned to bring the zero in any position.

Fig. 178 shows a vernier bevel protractor, which is used for checking angles. Toolmakers' buttons, which are used in locating holes for boring, are shown in Fig. 179, with a sectional view showing the method of attaching a button to the

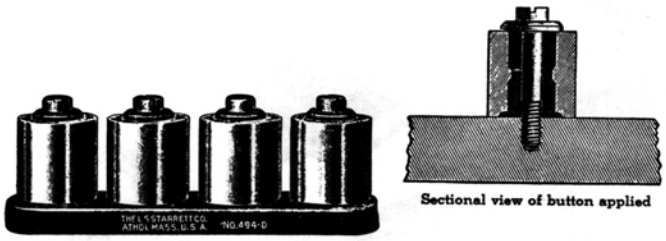


Fig. 179. Toolmakers' Buttons

work. Fig. 180 illustrates the application of these buttons.

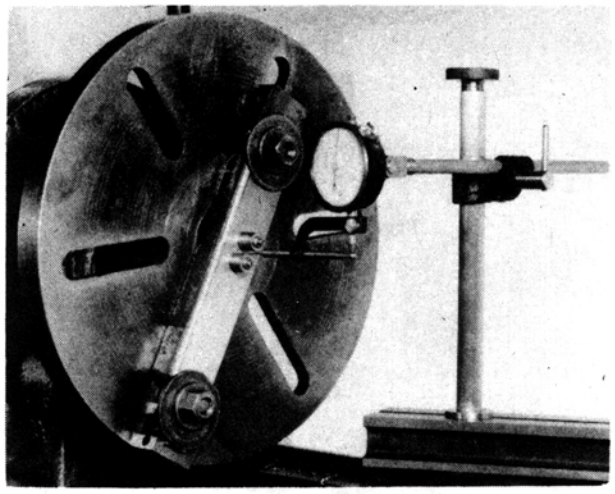


Fig. 180. Application of Toolmakers' Buttons

The ring gages in Figs. 181 and 183 are used for external measuring and the plug gages in Figs. 182 and 184 are used for internal measuring. Fig. 185 shows outside and inside caliper gages.

Fig. 186. Bench Centers

The bench centers in Fig. 186 are used for checking work mounted on centers. Fig. 187 shows a set of thickness gages. These are used for measuring the space between two surfaces.

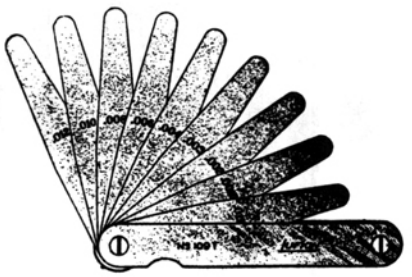


Fig. 187. Thickness Gage

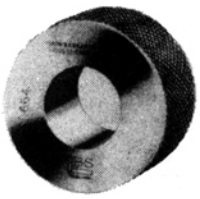


Fig. 181. Ring Gage

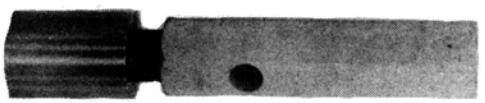


Fig. 182. Plug Gage

Fig. 183. Ring Thread Gage

Fig. 184. Plug Thread Gage

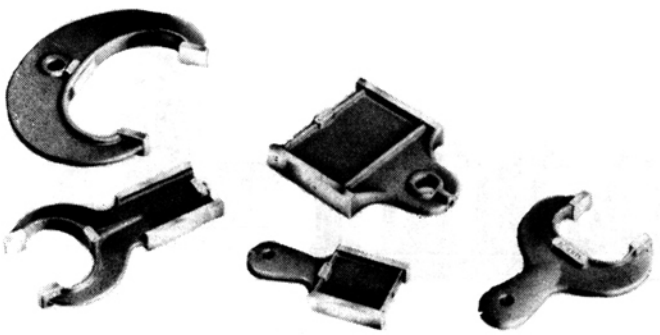


Fig. 185. Outside and Inside Caliper Gages

Fig. 188 shows a telescoping gage, which is used for internal measuring, as

Fig. 188. Telescoping Gage

illustrated in Fig. 189 (this illustration shows the gage being used on a blanking die). Fig. 190 shows the method of taking the measurement from a telescoping gage with a micrometer.

Fig. 195 is used for checking and setting 60° threading tools, while the gages in Figs. 196 and 197 are used for grinding and setting threading tools for cutting Acme threads and

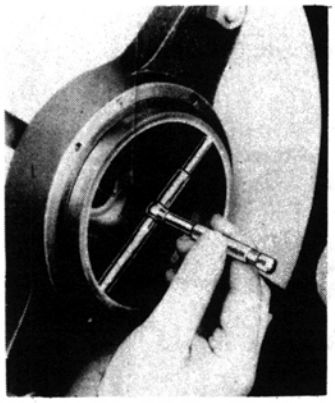


Fig. 189. Telescoping Gage in Use on a Die

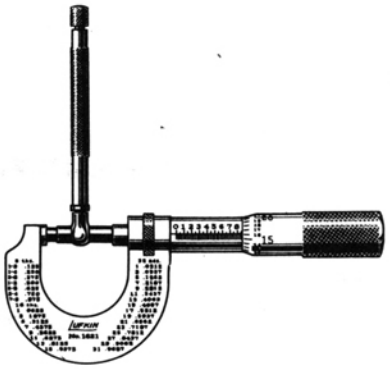


Fig. 190. Taking the Measurement of a Telescoping Gage

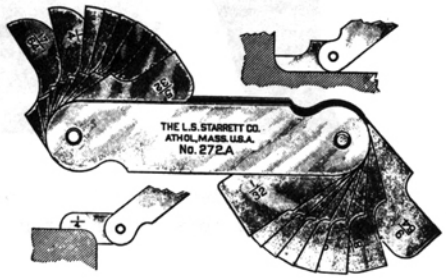


Fig. 191. Radius Gage

The radius and fillet gage in Fig. 191 consists of a number of steel blades marked in fractions of an inch, each blade having the corresponding radius accurately formed in it. The gage is used to check male and female radii, a female radius being called a fillet (see Figs. 192 and 193).

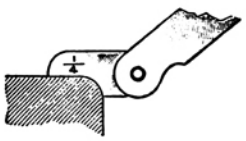


Fig. 192
Checking a Radius

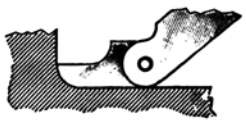


Fig. 193
Checking a Fillet

Figs. 194 to 199 illustrate several gages used in checking threads and threading tools. The screw pitch gage in Fig. 194 has a series of blades which are accurately notched and numbered according to the measure of the thread pitches in common use. It is used for checking the number of threads per inch of screws and nuts. The center gage in

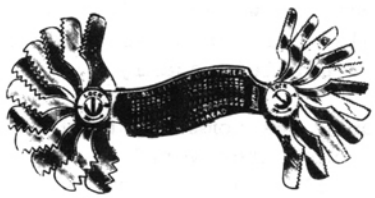


Fig. 194. Screw Pitch Gage

Fig. 195. Center Gage

Fig. 196. Acme Thread Tool Gage

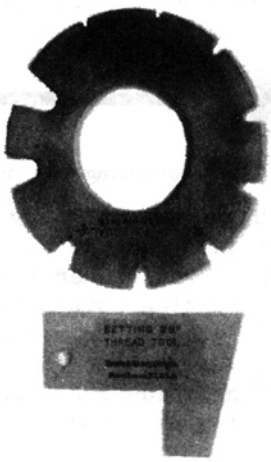


Fig. 197. Worm Thread Tool Gage

worm threads. Fig. 198 shows the method used in checking threads with an ordinary micrometer, while Fig. 199 shows the screw thread micrometer for measuring threads.

Fig. 198. Checking threads with three wires and common micrometer

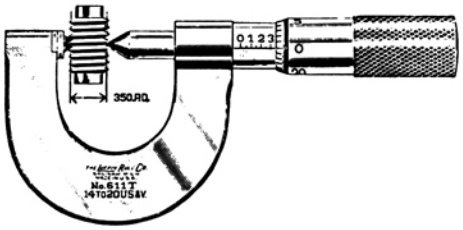


Fig. 199. Screw Thread Micrometer

37. What are the principal tools used in soldering operations?

A. Figs. 200 to 203 show soldering coppers, the square pointed copper being the one used for general work. The electric soldering copper is heated by electricity. The flat file

Fig. 200. Square Point Copper

Fig. 201. Bottoming Copper

in Fig. 204 is used for abraiding or smoothing metal, and is fitted into the file handle shown in Fig. 205. The tinners' snips in Fig. 206 are used for cutting soft metals. The ladle (Fig. 207) is used for melting solder or other materials. The blow torch in Fig. 208 is used for heating soldering coppers, melting materials in ladles (though a different

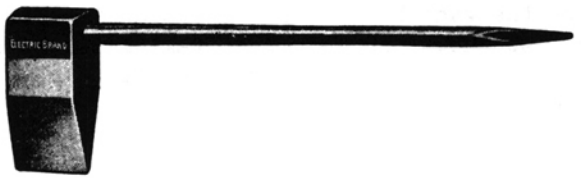


Fig. 202. Hatchet Copper

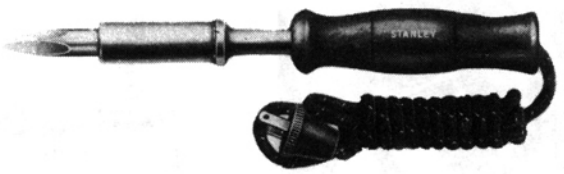


Fig. 203. Electric Copper

Fig. 204. Flat File

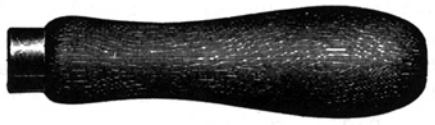


Fig. 205. File Handle

Fig. 206. Tinners' Snips

Fig. 207. Ladle

type of torch, known as a fire pot, is preferable for this), or for other needs in which heat must be applied locally.

38. What tools are commonly used in woodworking and carpenter work?

A. Figs. 209 and 210 show two folding

Fig. 208. Blow Torch

wood rules, the boxwood rule and the well-known zig-zag rule, both used for general measuring by the person working with wood.

The hand saw in Fig. 211 is the type generally used by carpenters. The compass saw (Fig. 212) is used for making circular holes. The level shown in Fig. 213 is used in testing horizontal and vertical surfaces.

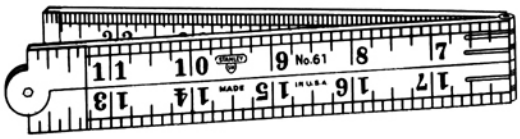


Fig. 209. Boxwood Folding Rule

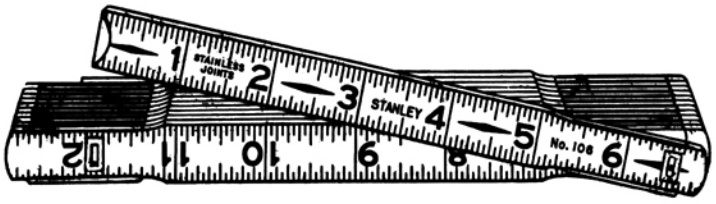


Fig. 210. Zig-zag Rule

Fig. 211. Hand Saw

Fig. 212. Compass Saw

Fig. 213. Wood Level

The marking gage in Fig. 214 is used for scribing lines parallel to an edge. The iron plane (Fig. 215) is used for producing a

Fig. 214. Marking Gage

Fig. 215. Iron Plane

Fig. 216. Drawing Knife

smooth finish on wood, and the drawing knife (Fig. 216) is used for peeling bark and roughing out narrow surfaces. The spoke shave (Fig. 217) is used for roughing out cylindrical wood work, and

Fig. 217. Spoke Shave

the flat wood chisel (Fig. 218) is used for cutting slots, etc. The wood turning tools in Fig. 219 are used when a piece of wood is turned in a lathe.

Fig. 218. Flat Wood Chisel

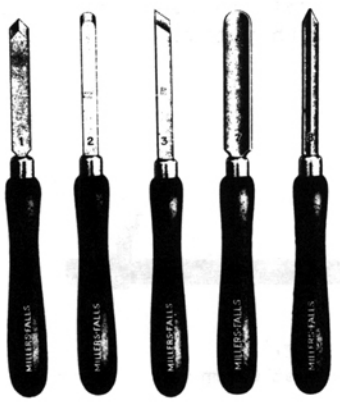


Fig. 219. Wood Turning Tools

Fig. 220 shows a bit brace, used for revolving bits when drilling in wood. Two types of bits are shown in Figs. 221 and 222. Fig. 223 shows a plumb bob, which is used to establish vertical lines. The steel square in Fig. 224 is used by carpenters for measuring and squaring work.

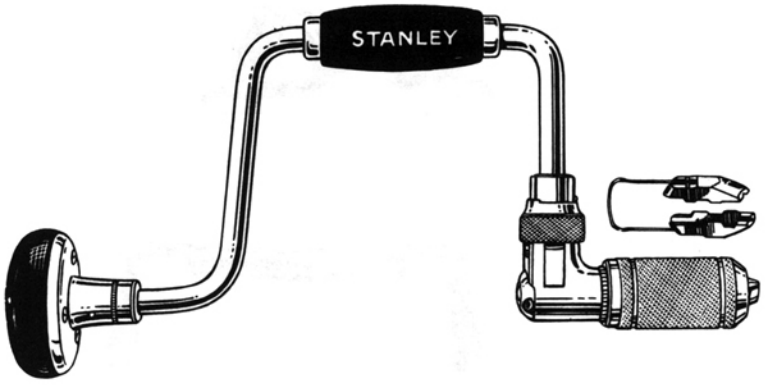


Fig. 220. Bit Brace

Fig. 221. Auger Bit

Fig. 222. Expansion Bit

The cabinet clamp in Fig. 225 is used for clamping glued joints while they are drying. The corrugated steel fastener in Fig. 226 is used as a permanent joint reenforcement, while the pinch dog of Fig. 227 is used simply for drawing glued joints together. Glass is cut with the cutter shown in Fig. 228. The fillet sticker in Fig. 229 is used for setting wax fillets, and is heated by the alcohol lamp shown in Fig. 230. The putty knife (Fig. 231) is used for filling holes and joints with putty.

Fig. 223. Plumb Bob

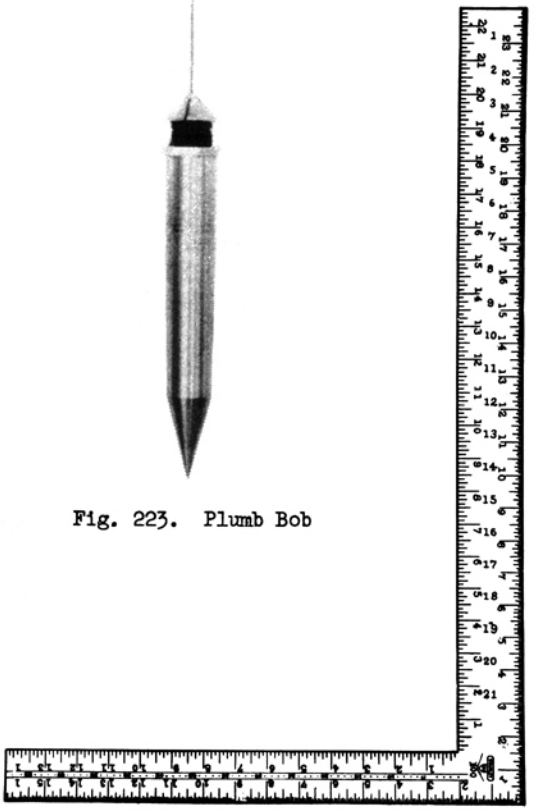


Fig. 224. Carpenters' Steel Square

Fig. 228. Glass Cutter

Fig. 229. Fillet Sticker

Fig. 225
Cabinet Clamp

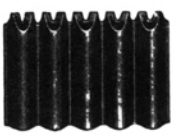


Fig. 226
Steel Fastener

Fig. 227
Pinch Dog

Fig. 230
Alcohol Lamp

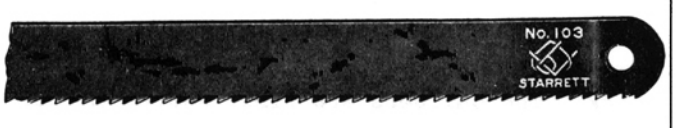
Fig. 231. Putty Knife

39. What is a hack saw? Why is it necessary to have different kinds of hack saw blades?

A. Fig. 232 shows a hand hack saw frame, used to hold hack saw blades while cutting metal. Blades having teeth of various sizes are used, according to the kind of metal being cut. Figure 233 shows three blades, with 10, 18, and 32 teeth. The 10-tooth blade is a heavy blade with coarse teeth. It can be used only in a power saw, as it is too heavy for use in a hand frame. The 18-tooth blade is an all-around saw for general work (aluminum, babbitt, tool steel, high speed steel, cast iron, etc.). The 32-tooth blade is recommended for fine stock, tubing, sheet metal, etc.

Fig. 232. Hand Hack Saw Frame

10-Tooth Blade



18-Tooth Blade

32-Tooth Blade

Fig. 233. Hack Saw Blades

Different kinds of steel are used in making hack saw blades for different classes of work, tool steel, high speed steel, or tungsten alloy steel being generally used. A blade should be placed in the frame with the teeth pointing forward, so that the forward stroke will be the cutting stroke, and the blade should be drawn tight enough so that it will not bend.

40. Describe a divider and explain how it is used.

A. A divider (Fig. 234) is a tool for measuring the distance between points, for transferring a distance directly from a rule, and for scribing circles and arcs (Fig. 235). It consists of two legs hardened on the ends and usually has a spring adjustment. The size is determined by the length of a leg from the pivot to the point.

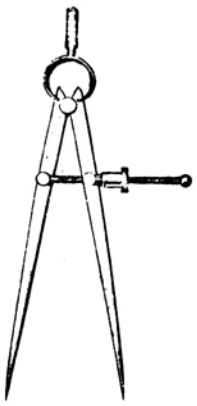


Fig. 234

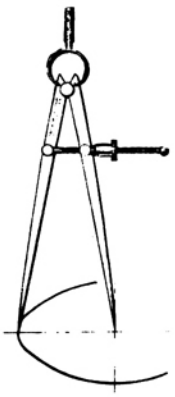


Fig. 235

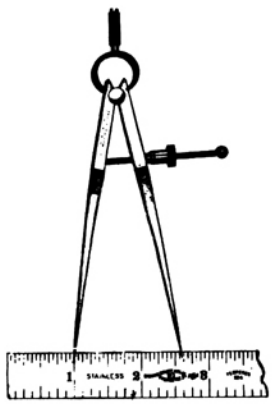


Fig. 236

In scribing a circle, first set the divider to the radius of the circle desired, as shown in Fig. 236. Then set one leg of the divider in the center punched hole and scribe short arcs on opposite sides of the center. If the distance between arcs is not equal to the required diameter, make the necessary adjustment and scribe the circle.

NOTE: When using a divider on finished work, its surface should first be covered with copper sulfate (blue vitriol) or layout ink (a solution of gentian violet, orange shellac, and alcohol), so that the layout lines will be clearer. Unfinished work should be covered with chalk or with a layout paint made of calcimine.

41. Describe a trammel and explain how it is used.

A. A trammel is a tool used to measure the distance between points too great to be reached with an ordinary divider. It consists of two adjustable points on a bar. In setting the trammel (shown in Fig. 237) to a given size, the points are first set approximately and the final adjustment made with the thumb screw. A rule is laid flat on the work bench, one point of the trammel is placed on a selected mark on the rule, and the other point is adjusted until it is set at the required dimension.

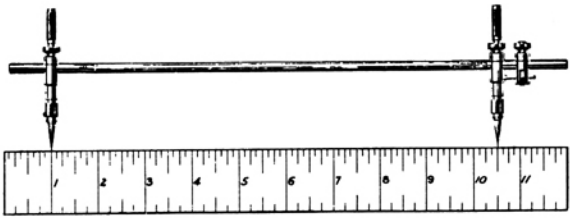


Fig. 237

The method of scribing a circle is shown in Fig. 238. Before scribing a complete circle, one should scribe short arcs on the center line on opposite sides of the center. If the distance between these arcs is the same as the diameter of the required circle, the setting is correct and no further adjustment is necessary.

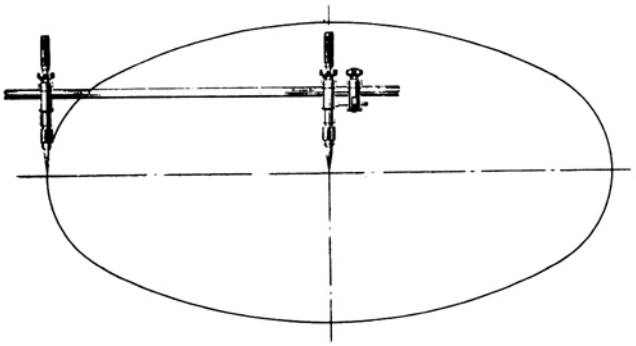


Fig. 238

42. Describe outside calipers and explain how they are used.

A. Calipers are used for measuring distances between or over surfaces, or for comparing distances or sizes with standards. Calipers have two adjustable legs which work on a hinged joint. Outside calipers have curved legs for measuring outside dimensions (Fig. 239).

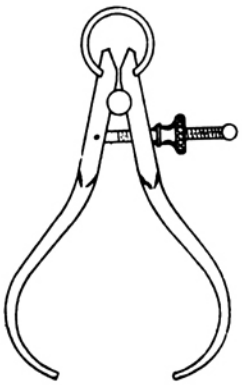


Fig. 239

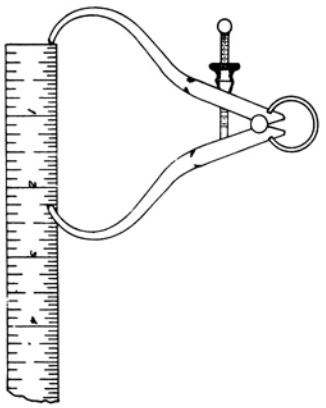


Fig. 240

In setting outside calipers with a rule, first set the calipers at approximately the given size. Then, with one point resting on the end of the rule, adjust the calipers until the other point is opposite the proper mark on the rule (Fig. 240). To insure an accurate setting the center line of the points must be parallel to the edge of the rule.

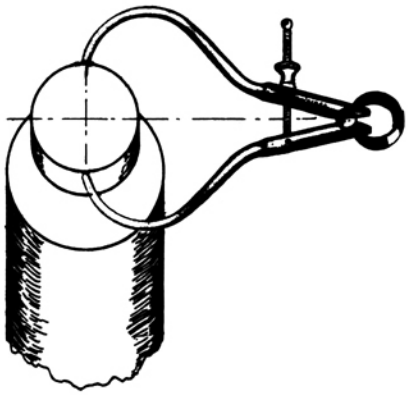


Fig. 241

Fig. 241 shows how outside calipers are used to measure the diameter of round stock. The calipers are first set to the approximate diameter of the stock. Then they are held at right angles to the center line of the work and moved back and forth across the center line while they are adjusted until the points bear lightly on the work. This is called "getting the feel." When the tool has been adjusted properly, the diameter may be read as indicated in Fig. 240.

43. Describe inside calipers and explain how they are used.

A. Inside calipers are calipers with legs curved for measuring the diameter of holes, the distance between two surfaces, the width

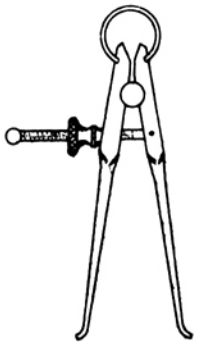


Fig. 242

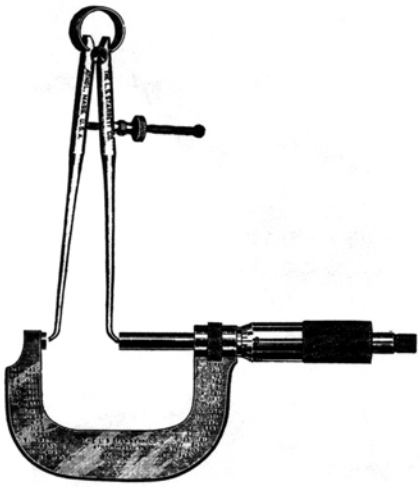


Fig. 243

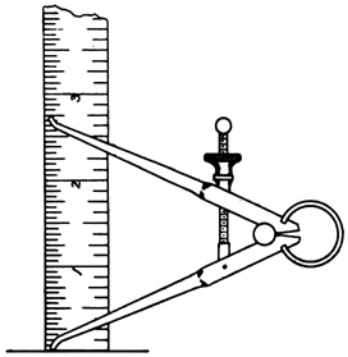


Fig. 244

of slots, and other similar jobs (Fig. 242).

In setting inside calipers, first set a micrometer to the required dimension. Then hold the micrometer and calipers in the position shown in Fig. 243 and adjust until the proper "feel" is secured. Another method is to hold a rule at right angles to a flat surface, set one leg of the inside calipers on the flat surface, and adjust until the other leg is opposite the proper mark on the rule (Fig. 244 shows this method of setting the calipers).

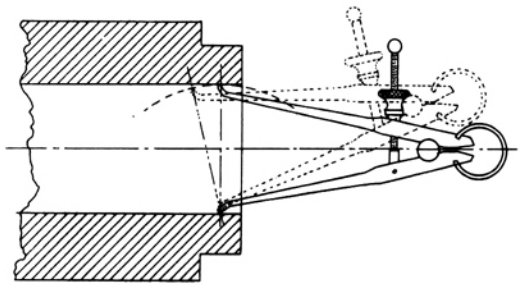


Fig. 245

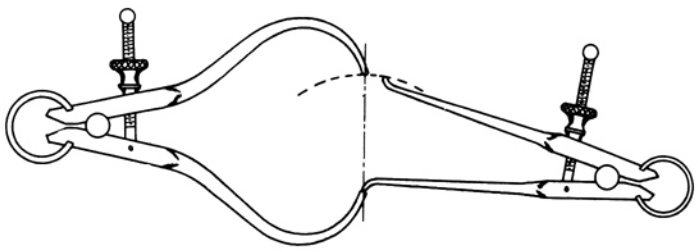


Fig. 246

To measure the diameter of a hole with inside calipers, first set the calipers approximately to the size of the hole. Then hold one leg against the wall of the hole and adjust the other leg until it just touches a point diametrically opposite, as illustrated in Fig. 245. The next step is to transfer the measurement from the inside calipers to a micrometer. Great care must be taken in adjusting the calipers and transferring measurements. With a little practice a mechanic should "caliper" a hole within one-half a thousandth of an inch.

44. Describe the transferring of a measurement from outside to inside calipers.

A. When a measurement has to be transferred from outside to inside calipers, both calipers are held so that they are in the position shown in Fig. 246. With one extreme point of a leg of the inside calipers placed on the extreme point of a leg of the outside calipers, adjust the inside calipers until the two extreme points touch lightly. Care must be taken not to "force" either pair of calipers or a true reading will not be obtained.

45. Describe hermaphrodite calipers and explain how they are used.

A. Hermaphrodite calipers have two legs which work on a hinge joint (Fig. 247). One leg is similar to a leg on a pair of dividers and the other is similar to a leg on a pair of outside calipers. Hermaphrodite calipers may be used to scribe arcs, as shown in Fig. 248, or as a marking gage in layout work, as shown in Fig. 249.

Fig. 247

Fig. 248

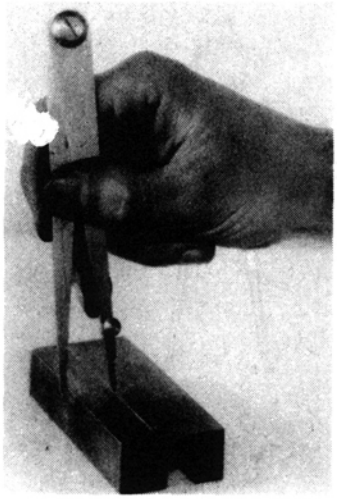


Fig. 249

To set hermaphrodite calipers to a rule, adjust the scriber leg until it is slightly shorter than the curved leg. Then, with the curved leg set on the end of a rule, adjust the scriber leg to a point opposite the required line on the rule, as illustrated in Fig. 250.

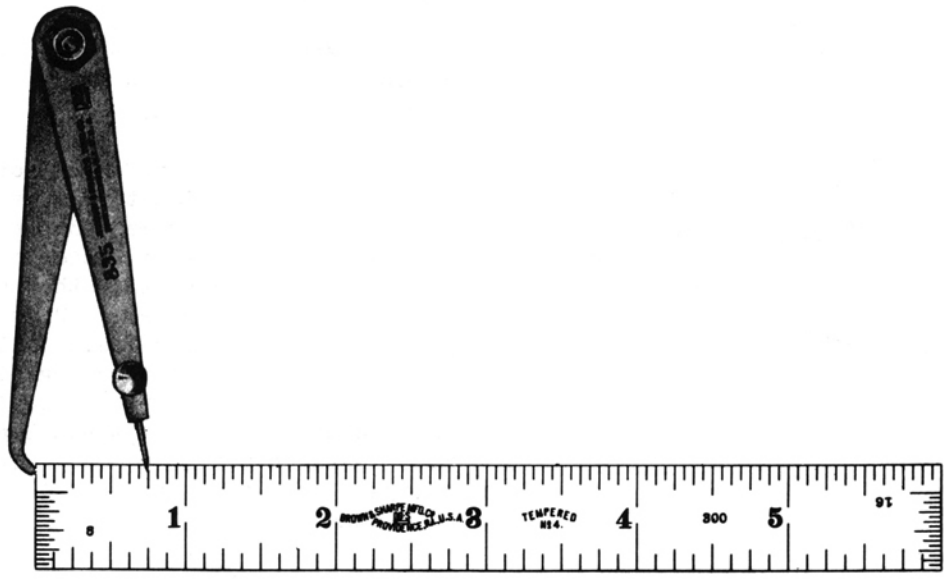


Fig. 250

Chapter 4

RULES

1. What is the difference between a rule and a scale?

A. A rule is a graduated measuring instrument, made of wood, metal, or other suitable material (see Fig. 251). It is usually graduated to indicate inches and fractions of an inch (or centimeters and millimeters). A scale is similar in appearance to a rule, as its surface is graduated into regular spaces, but these spaces differ decidedly from those on a rule, because they are larger or smaller than the actual measurements indicated. A scale therefore gives proportional measurements instead of the accurate measurements obtained with a rule. Figs. 252 and 253 illustrate two scales regularly used in the drawing room for measuring and laying out distances and dimensions.

2. Name some rules commonly used in the shop.

A. There are several different kinds of rules, each being especially adapted to certain classes of work. By choosing the proper rule it is easier for a mechanic to obtain accurate measurements. In addition to the well-known flexible rule shown in Fig. 251, which is used in many ways in the shop, two other rules commonly used are shown in Figs. 254 and 255. The narrow rule is used for work on which other rules are too wide and the hook rule is used where the hook is an advantage in measuring.

3. Briefly describe and tell the purpose of a shrink rule.

A. When molten metal is poured into a mould and allowed to cool and solidify it will contract. To make allowance for this contraction or shrink and to maintain exact relationship for all dimensions, the pattern maker uses a shrink rule. Since the rate of contraction changes with different materials each shrink rule has the shrink allowance stamped on it. The shrink rule, Fig. 256, is actually 12-1/8 inches long, the additional length being taken up proportionally throughout its length.

Fig. 251. Six-Inch Flexible Rule

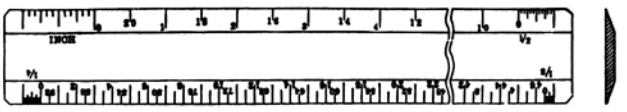


Fig. 252. Flat Boxwood Scale

Fig. 253. Triangular Boxwood Scale

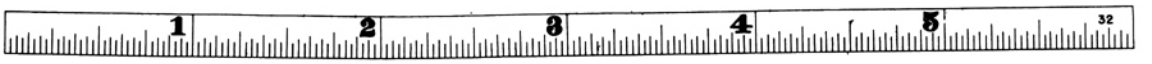


Fig. 254. Narrow Rule

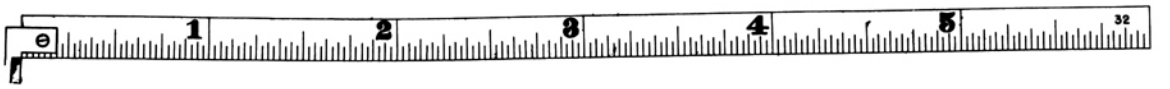


Fig. 255. Hook Rule

Fig. 256. Shrink Rule

4. How is a six-inch rule usually graduated?

A. As every mechanic uses a rule frequently (Fig. 257 shows one application) it is important to understand this type thoroughly.

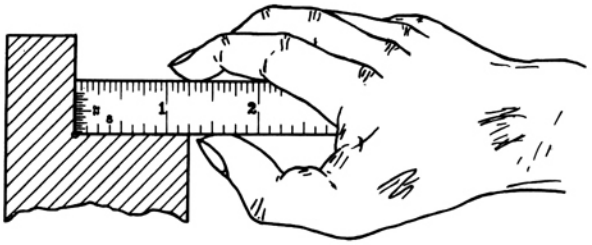


Fig. 257. Using a Rule

One of the most frequent ways of graduating this rule is to divide the inches into halves, quarters, eighths, sixteenths, thirty-seconds, and sixty-fourths. One side of the rule will then usually have coarse graduations, 8ths on one edge and 16ths on the other (see Fig. 258), while the other side

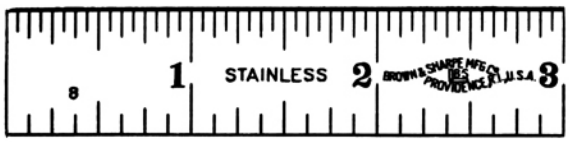


Fig. 258. Rule Graduated in 8ths and 16ths

will have fine graduations, 32nds on one edge and 64ths on the other (see Fig. 259). Since the Ford Motor Company has adopted the decimal system for specifying dimensions on all new designs, the rule in general use is graduated in 32nds and 64ths on one side and in 10ths and 50ths on the other side (see Fig.

Fig. 259. Rule Graduated in 32nds and 64ths

263). Various other graduations and combinations are of course frequently used, and any desired marking will be furnished by a rule manufacturer on a special order.

5. What is the best method of learning to read a rule?

A. The following procedure is recommended:

(1) Learn the 8ths and 16ths shown in Fig. 258 (or the 10ths and 50ths if a rule is so graduated).

(2) Become familiar with the 32nds and 64ths (Fig. 259).

(3) Thoroughly understand the readings shown in Fig. 260.

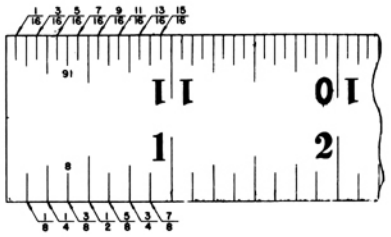


Fig. 260

(4) Practice until measurements such as those in Figs. 261 and 262 can be read quickly.

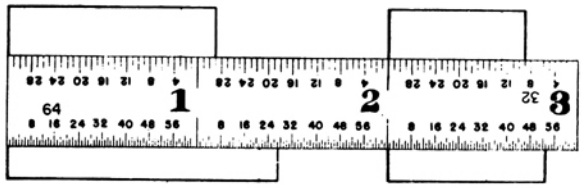


Fig. 261

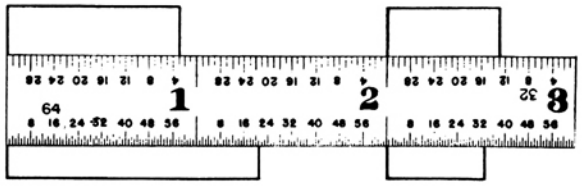


Fig. 262

6. Explain the decimal system used by the Ford Motor Company for specifying dimensions.

A. The Ford Motor Company has adopted the decimal system for specifying divisions of an inch, this system replacing the old one which used common fractions.

It is obvious that rules graduated for the new system must be used in working on jobs dimensioned in decimals. Fig. 263 shows a six-inch flexible rule graduated in 10ths and 50ths.

Fig. 263. Six-Inch Rule Graduated in 10ths and 50ths

The system is effective on all new drawings but is not used for changes on drawings showing the old style dimensions. The following outline explains the new system, which saves time and makes calculating easier.

Instead of dimensioning drawings 1/2, 1/4, 1/8, 1/16, 1/32, and 1/64 of an inch, dimensions on drawings of new parts are specified in tenths or hundredths of an inch. Thus, instead of such dimensions as 1-5/16, 2-7/32, 5-33/64 and 6-1/2 on drawings, the mixed decimals 1.32, 2.22, 5.52, and 6.5 are used (see Fig. 264).

It is understood that one or two place decimals carry the same degree of accuracy that the common fractions carried previously, namely, + or - .010 on a finished drawing. When greater accuracy than this is demanded, a three or four place decimal is used and a definite tolerance is specified as .400 - .402.

The use of an odd number for the second place in a dimension of a diameter is to be avoided, because "halving" the diameter to obtain the radius will give a three place number. An effort should therefore be made to keep the second figure even. This of course does not apply to ordinary dimensions which will not be "halved."

1. List and find the combined length of the lettered divisions in Figs. 265(a) and (b).

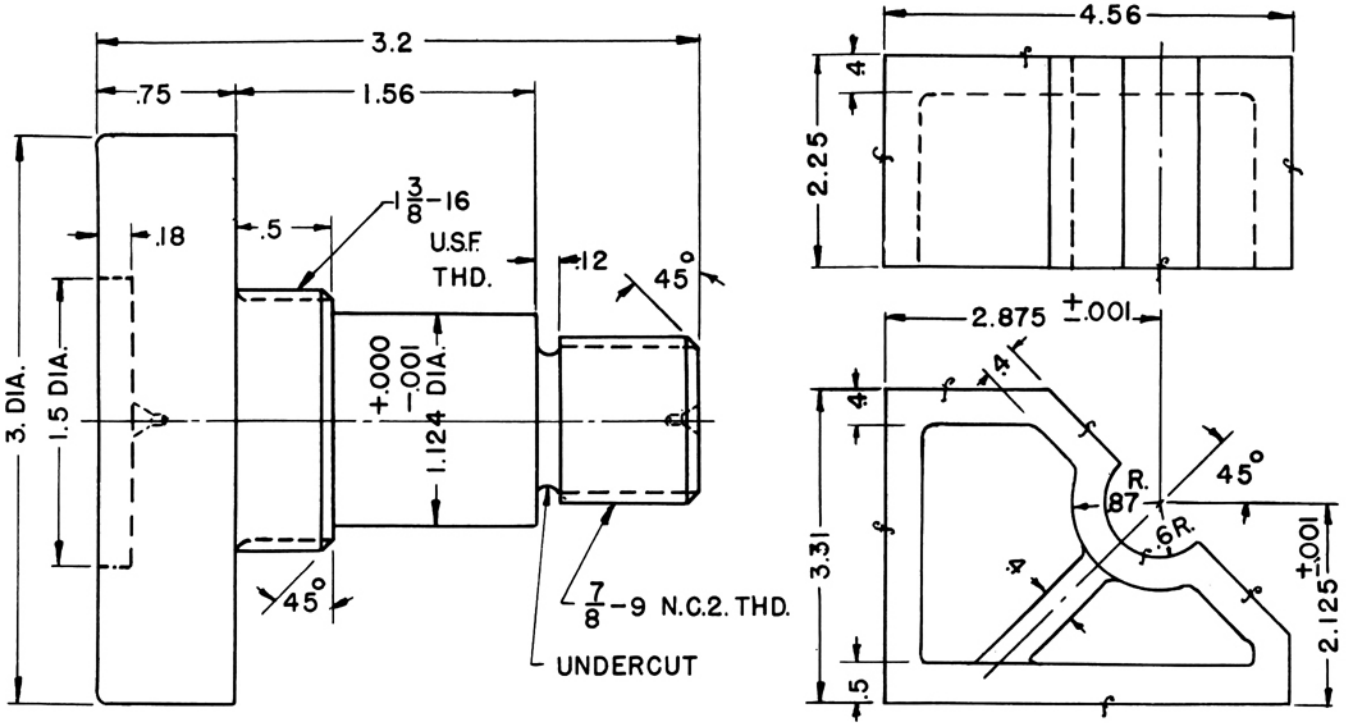


Fig. 264. Decimal System of Dimensioning

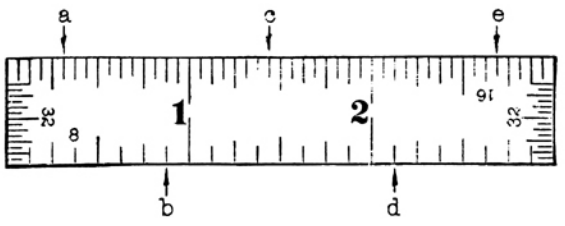


Fig. 265(a)

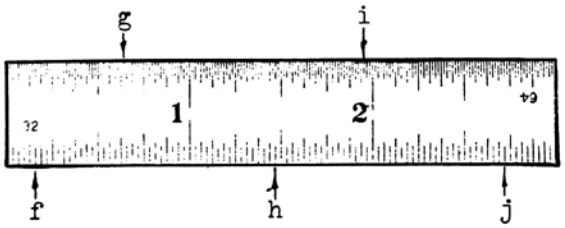


Fig. 265(b)

2. Measure the lines below and list their lengths. Find their sum,
   (a)_____(b)_____ (c)_____
   (d)_____ (e)_____
   (f)__ (g)_____ (h)_____
   (i)_____
   (j)_____

3. Draw lines as near the following lengths as possible.

(a) 3-3/4	(e) 2-7/16	(h) 1.3750
(b) 1-1/8	(f) .0312	(i) .453125
(c) 3/16	(g) .1875	(j) 3.765625
(d) 1/2		

4. Measure the lettered dimensions in the following sketches of Fig. 266 and write them in the proper order.

5. Draw a sketch of a rectangular bar 1-3/8 x 2-5/16 x 4-11/32.
   Draw a sketch of a rectangular bar 1-1/4 x 1-7/16 x 1-1/2.
   Draw a sketch of a bar 1.5 x 3.87.

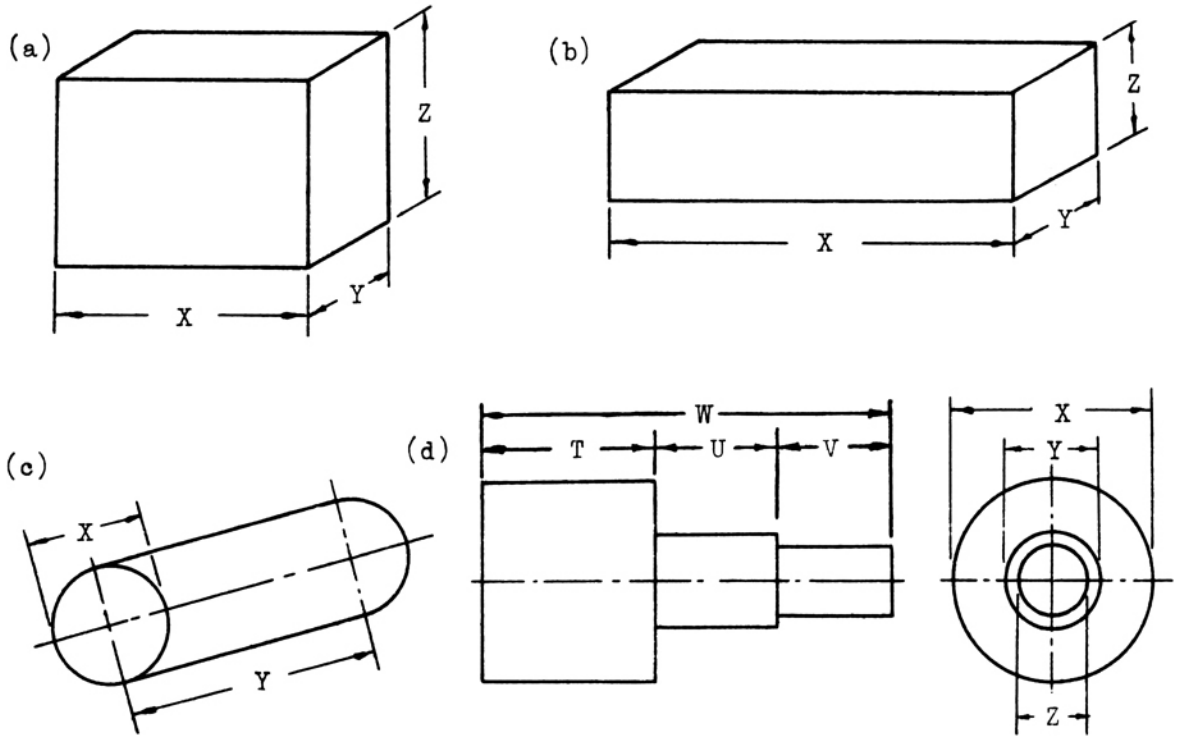


Fig. 266 (a, b, c, and d)

Chapter 5

MICROMETERS

The micrometer is the most commonly used precision tool in the toolroom, used for accurately measuring dimensions. In order to use such a tool effectively and appreciate some of the mechanical principles on which it works, its origin, history, construction, use, and care should be understood.

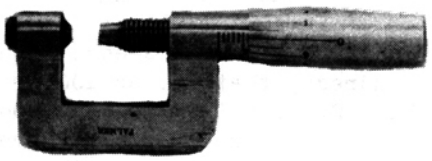


Fig. 267. Micrometer Caliper of 1848

The micrometer (see Fig. 267) was invented by Jean Palmer, a Frenchman, in 1848. It was not introduced into this country until 1867, when J. R. Brown and L. Sharpe brought back a Palmer micrometer with them after a visit to the Paris Exposition. From this instrument they developed a micrometer which was the beginning of our modern micrometer. This micrometer was introduced in 1877 (see Fig. 268), and was the first one of this type to be sold in the United States. Fig. 269 shows an improved micrometer introduced in 1885 by the same concern. This brief history of the micrometer shows that it is a comparatively new tool. It has been improved from time to time by different persons, until the modern tool in Fig. 271 is the result.

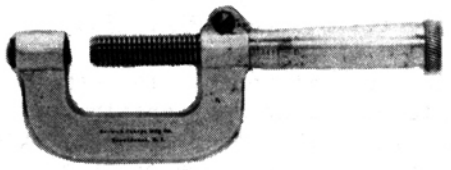


Fig. 268. Micrometer Caliper of 1877

Three different types of micrometers are used in the shop. These are the outside micrometer of Fig. 271 (including the thread micrometer in Fig. 275), the depth micrometer of Fig. 270, and the inside micrometer shown in Fig. 272.

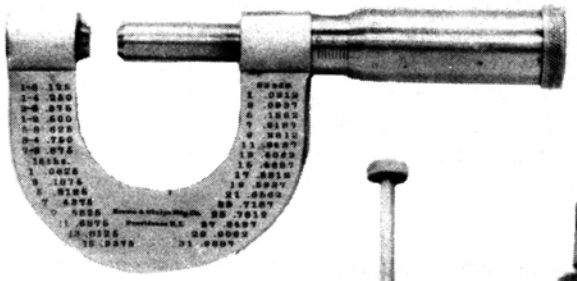


Fig. 269. Improved Micrometer Caliper of 1885

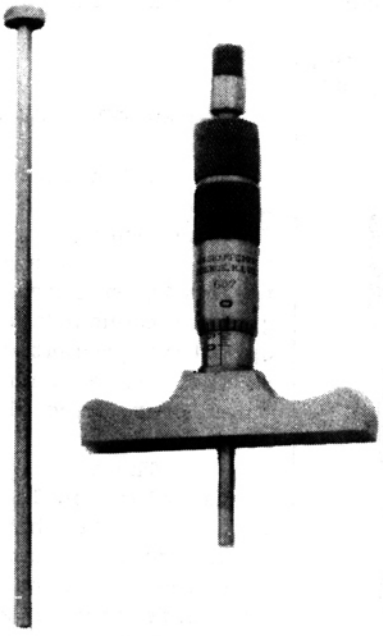


Fig. 270

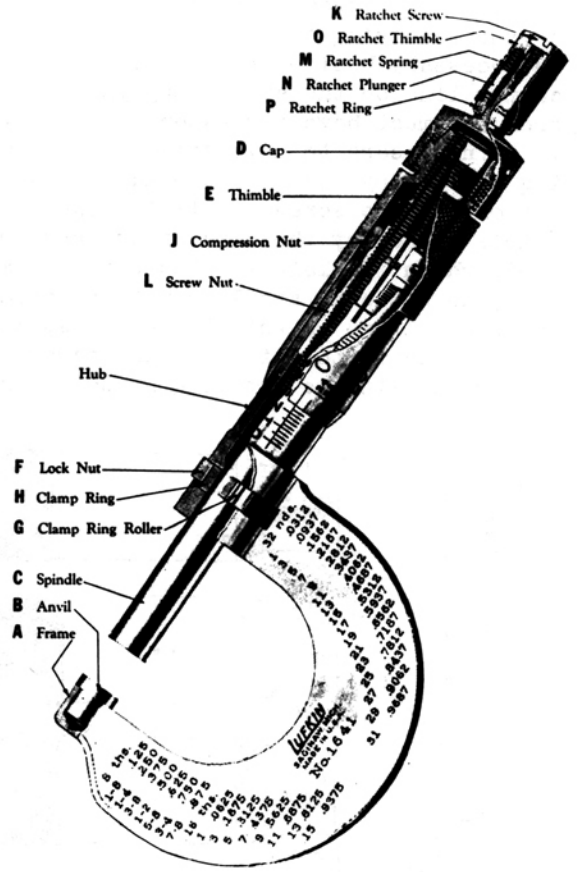


Fig. 271

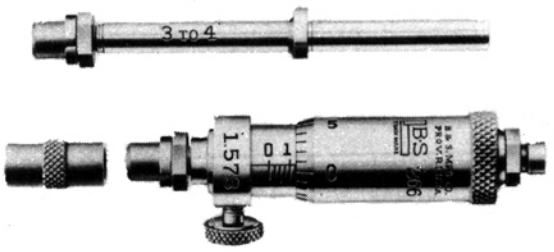


Fig. 272

1. What are the five principal parts of a micrometer?

A. The frame, anvil, spindle, sleeve or barrel, and thimble (see Fig. 271).

2. Explain how a micrometer is read.

A. A micrometer is very easily read, but of course, like many other things, rapid work is obtained only after some practice. When once learned it can be read at a glance.

The micrometer divides the inch into 1000 parts. As usually made, it has a 40 pitch screw (a screw with 40 threads per inch) which advances, through a nut, .025 of an inch per revolution. It is evident that if only measurements of .025" or less were to be made, all the graduating could be on the end of the revolving thimble, and all that would be necessary besides this would be an indicating line on the stationary part. As a micrometer must have a greater range, however, it is necessary to have some means of counting and adding together the additional revolutions of the screw. This is done in a very simple manner by the graduations and numbers used, which is plainly illustrated in Fig. 273 (the sketches are larger than actual size). The cross lines on the sleeve are spaced .025" apart to equal the pitch of the screw. A revolution line is cut lengthwise on the sleeve, and in connection with the zero line on the thimble, counts whole revolutions of the screw. When the end of the thimble matches any of the cross lines and the zero line matches with the revolution line, the number of spaces exposed denotes the number of revolutions made. Every fourth cross line is numbered from 0 to 10.

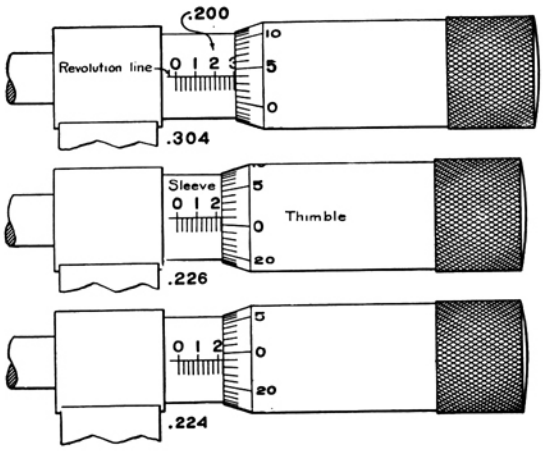


Fig. 273

In the first illustration of Fig. 273, the reading is .304", showing .300" on the sleeve and .004" on the thimble. In the second illustration the reading is .226" showing .225" on the sleeve and .001" on the thimble. The reading in the third illustration is .224", showing .200" on the sleeve and .024" on the spindle.

The figures on the sleeve should be taken as hundreds, that is, as 100, 200, 300, etc. In Fig. 273 the thimble is purposely shown close to the lines, as these are the points where a mistake is most likely to occur. In the .226" reading, while the end of the thimble may appear to match the cross line, it is evident that it does not, because the zero line on the thimble does not coincide with the revolution line on the sleeve. The zero line is one space advanced, which is of course added to the .225", making the reading .226".

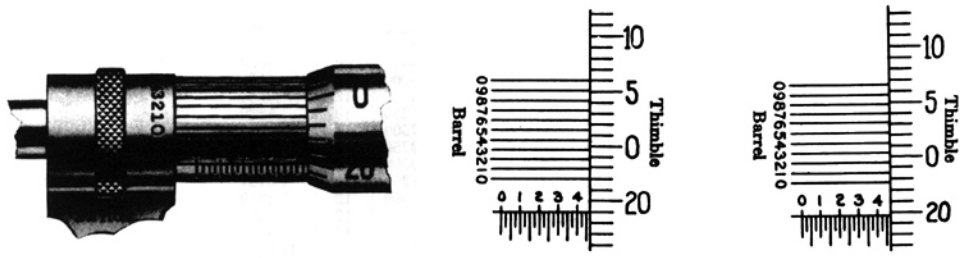
A very good way to learn to read this instrument is to take a one inch micrometer, turn it down to zero (or against the anvil), and then turn the screw back while you count the graduations on the thimble for four revolutions (which means counting these graduations to 100). The method of counting the cross graduating will then be easily understood.

READING THE MICROMETER IN TEN-THOUSANDTHS

While the thousandths part of an inch is the most convenient unit and is fine enough for the general run of machine work, there are times when this must be further divided. With the ordinary micrometer one-half and one-quarter thousandths are easily estimated. Micrometers graduated with a vernier arranged so that ten-thousandths can be read on the thimble are used where finer adjustments are required.

The vernier used consists of ten divisions which equal, in over-all space, nine divisions on the thimble. Thus, one division on the vernier equals .1 of .009" or .0009". Each graduation on the thimble equals .001". The difference in space between a division on the thimble and a division on the barrel is equal to .0010" - .0009" or .0001". The two

zero lines on the vernier coincide with lines on the thimble when the reading is exact with respect to the number of thousandths. The difference between the lines on the thimble and lines on the vernier at number 1 is .0001", at number 2 is .0002", at number 3 is .0003", etc. For example, when the 1st, 2nd, or 3rd lines coincide, the thimble has moved past the zero setting .0001", .0002", or .0003" to bring these lines together.



(a) (b) (c)

Fig. 274

To Read: First obtain the reading for thousandths as described in the preceding section and then add the ten-thousandths which are indicated by the line on the vernier that coincides with a line on the thimble.

Example: In Fig. 274(b) the reading is .4690". There are no ten-thousandths to be added because the two zeros on the vernier coincide with lines on the thimble. In Fig. 274(c) the seventh graduation on the vernier coincides with a line on the thimble, indicating that .0007 should be added to the thousandths reading. The correct reading is .4690" + .0007" = .4697".

3. Briefly describe the screw thread micrometer and explain its purpose.

A. The screw thread micrometer (Fig. 275) is a tool for measuring the pitch diameter of threads. The spindle is ground conical to an included angle of 60° and is slightly flattened on the end. The anvil has a 60° groove and is set to align perfectly with the spindle. This anvil may be either fixed or swiveled. In measuring screw threads, the angle of the point and the sides of the V come in contact with the cut surface of the thread so that the reading of the micrometer indicates the pitch diameter or the full size of the thread less the depth of one thread.

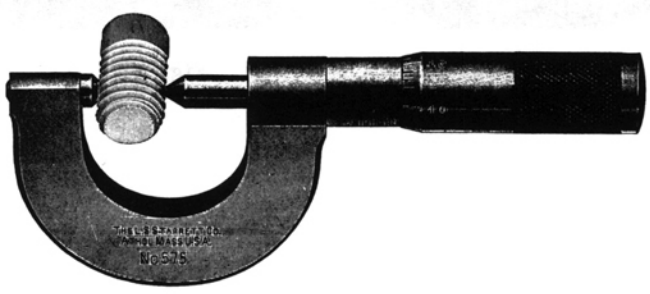


Fig. 275

The pitch line may be represented by a line drawn through the plane AB (Fig. 276) when the spindle and the anvil are in contact

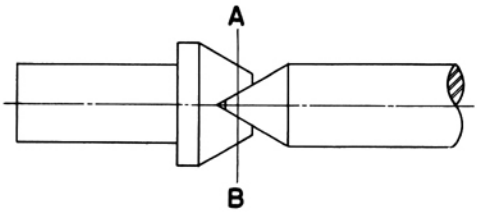


Fig. 276

with each other and the graduations on the thimble are set at zero. When the spindle is turned to open the micrometer the reading represents the distance between the pitch line on the anvil and the pitch line on the spindle, or the pitch diameter. Since the thread is measured from the angular surfaces, the actual outside diameter need not be considered. The thread micrometer is read the same as the ordinary outside micrometer.

There are two methods of measuring the pitch diameter of threads in common use, the thread micrometer, and the three-wire system. The thread micrometers are fast and convenient to use, but when set as described above, the reading is slightly distorted, the amount of distortion depending upon the helix angle of the thread being measured. While not as fast and convenient as the thread micrometer, the three-wire system is theoretically correct. The wires must be held very close to size because any error in the wire is multiplied when the measurement over the wires is calculated.

Pitch Diameter = Major (Outside) Diameter—Depth of Thread

COARSE THREAD SERIES				FINE THREAD SERIES			
Size		Threads Per Inch	Cal. Reading or Pitch Diam.	Size		Threads Per Inch	Cal. Reading or Pitch Diam.
Number or Fraction	Decimal	N	$D-\frac{.649519}{N}$	Number or Fraction	Decimal	N	$D-\frac{.649519}{N}$
1	.0730	64	.0629	0	.0600	80	.0519
2	.0860	56	.0744	1	.0730	72	.0640
3	.0990	48	.0855	2	.0860	64	.0759
4	.1120	40	.0958	3	.0990	56	.0874
5	.1250	40	.1088	4	.1120	48	.0985
6	.1380	32	.1177	5	.1250	44	.1102
8	.1640	32	.1437	6	.1380	40	.1218
10	.1900	24	.1629	8	.1640	36	.1460
12	.2160	24	.1889	10	.1900	32	.1697
1-4	.2500	20	.2175	12	.2160	28	.1928
5-16	.3125	18	.2764	1-4	.2500	28	.2268
3-8	.3750	16	.3344	5-16	.3125	24	.2854
7-16	.4375	14	.3911	3-8	.3750	24	.3479
1-2	.5000	13	.4500	7-16	.4375	20	.4050
9-16	.5625	12	.5084	1-2	.5000	20	.4675
5-8	.6250	11	.5660	9-16	.5625	18	.5264
3-4	.7500	10	.6850	5-8	.6250	18	.5889
7-8	.8750	9	.8028	3-4	.7500	16	.7094
1	1.0000	8	.9188	7-8	.8750	14	.8286
1 1-8	1.1250	7	1.0322	1	1.0000	14	.9536
1 1-4	1.2500	7	1.1572	1 1-8	1.1250	12	1.0709
1 1-2	1.5000	6	1.3917	1 1-4	1.2500	12	1.1959
1 3-4	1.7500	5	1.6201	1 1-2	1.5000	12	1.4459
2	2.0000	4 1-2	1.8557	1 3-4	1.7500	12	†1.6959
2 1-4	2.2500	4 1-2	2.1057	2	2.0000	12	†1.9459
2 1-2	2.5000	4	2.3376	2 1-4	2.2500	12	†2.1959
2 3-4	2.7500	4	2.5876	2 1-2	2.5000	12	†2.4459
3	3.0000	4	2.8376	2 3-4	2.7500	12	†2.6959
3	3.0000	3 1-2	2.8144	3	3.0000	10	†2.9350

The table of measurements shows the pitch diameters of American National threads for the coarse and fine thread series.

4. What are some of the applications of the common types of micrometers?

A. The 1" micrometer in Fig. 277 is used to measure the outside diameter of a small plug. In Fig. 278 a 2" micrometer is used to measure the diameter of stock in a lathe.

The methods of using a depth micrometer (Fig. 279) and an inside micrometer (Fig. 280) are shown below.

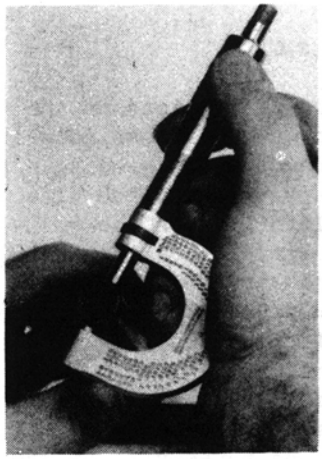


Fig. 277

Fig. 278

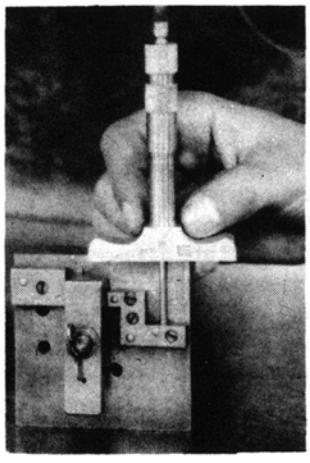


Fig. 279

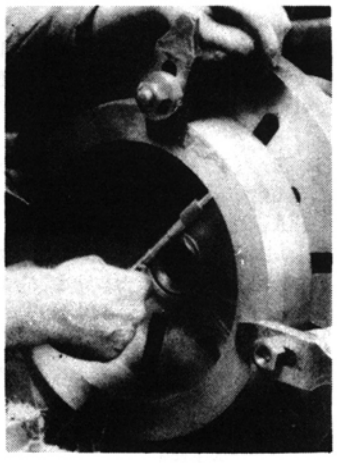


Fig. 280

Chapter 6

VERNIER GAGES

1. Describe the height gage.

A. The height gage (shown in Fig. 281) is a steel upright bar, usually 10" to 18" in height, and is graduated to read thousandths of an inch by means of a vernier scale on the movable jaw. The fixed jaw forms a base. The sharp point of the scriber is used to scribe lines on steel that has had the oxidized surface or scale removed.

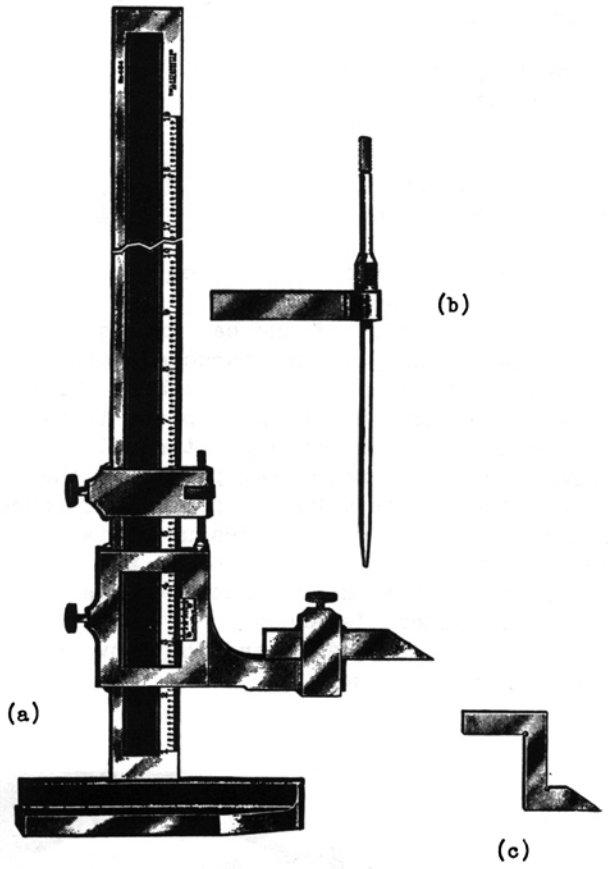


Fig. 281. (a) Height Gage, (b) H Attachment, (c) Offset Scriber.

Fig. 282 illustrates the vernier used, with a scale graduated into 40ths or .025ths of an inch (scale shown on height gage in Fig. 281). The vernier has 25 divisions which are numbered every fifth division, equaling, in total length, 24 divisions on the scale, or for length in inches,

$$24 \times \frac{1}{40} = 24 \times .025 = .600"$$

Thus one division on the vernier equals 1/25 of .600" = .024". Therefore the difference between a division on the vernier and a division on the scale on the bar is .025" - .024" = .001".

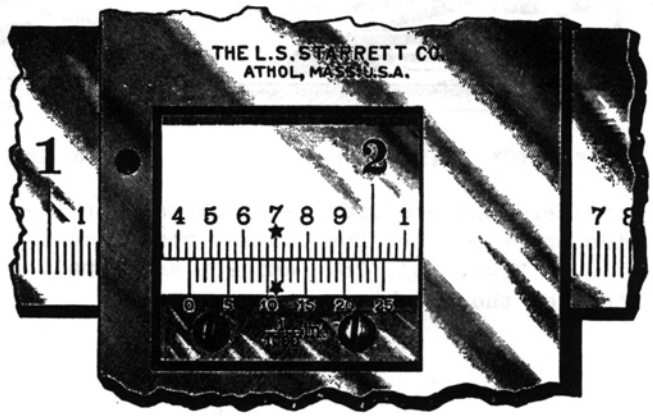


Fig. 282

To read the gage, note how many inches, how many tenths of an inch (or .100"), and how many fortieths of an inch (or .025") the zero mark on the vernier is from the zero mark on the bar. Then note the number of divisions on the vernier from its zero line to the line that exactly coincides with a line on the bar. For example, in Fig. 282 the vernier has been moved to the right one and four-tenths and one-fortieth inches (1.425"), as shown on the bar, and the eleventh line on the vernier coincides with a line on the bar (as indicated by the stars). Eleven thousandths of an inch is to be added to the reading on the bar, so that the total reading is one and four hundred thirty-six thousandths inches (1.436").

Care must be taken in reading the vernier to insure correct readings. Face the light with the vernier held in a horizontal position and tipped slightly so you can look directly down the lines on the vernier plate. In this way it can be more readily determined when a line on the vernier plate coincides with a line on the bar. A magnifying glass should be used so that the reading will be more accurate and cause less strain on the eyes.

NOTE. Be sure that the reading is taken on the same side of the bar each time.

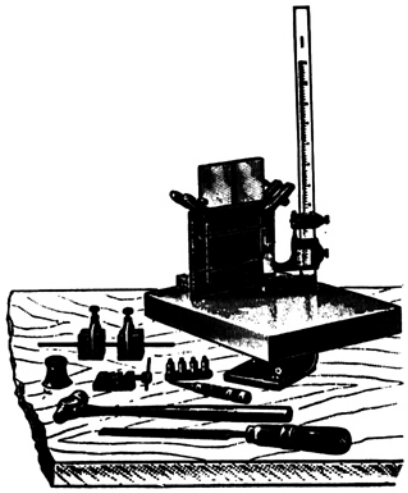


Fig. 283. Using Scriber

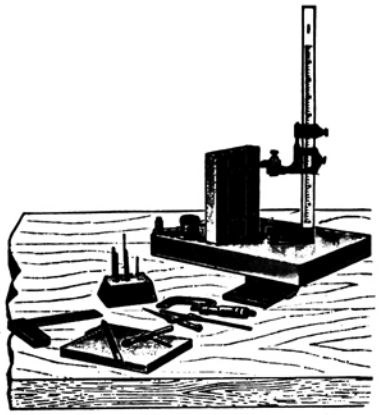


Fig. 284. Measuring Height

Remember that one side is graduated for inside and the other for outside measurements.

Figs. 283, 284, and 285 show applications of the height gage.

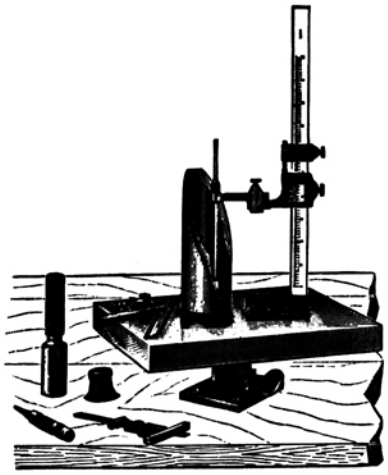


Fig. 285. Measuring Depth

2. Briefly describe the vernier caliper and explain its use.

A. The vernier caliper (Fig. 286) is a tool for checking inside and outside measurements. Both sides of the bar are usually graduated. The jaws are hardened, ground, and lapped parallel with each other. With the jaws in contact, the vernier plate is set on "0" on one side and at a point equal to the thickness of the measuring points on the other side. This makes it possible to check either inside or outside measurements without making any calculations. Points are placed on the bar and slide so that dividers may be set to transfer distances.

Figs. 287, 288, and 289 show the vernier caliper in use.

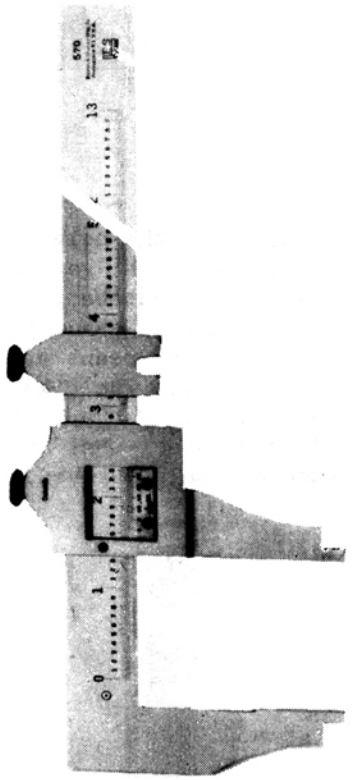


Fig. 286

Fig. 287
Checking External Dimension

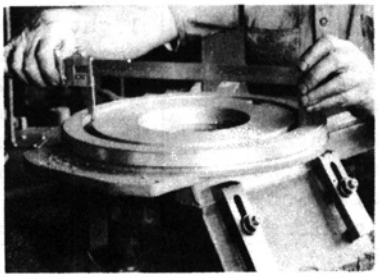


Fig. 288
Checking Diameter

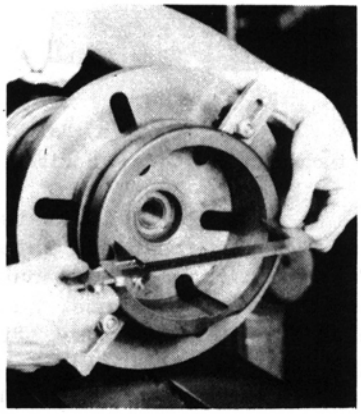


Fig. 289
Checking Inside Diameter

3. Describe the gear tooth vernier caliper.

A. The gear tooth caliper shown in Fig. 290 is designed to measure the thickness of a gear tooth on the pitch circle (chordal thickness) and the distance from the top of the tooth to the chord (corrected addendum), at the same time. The vernier on the gear tooth caliper is read in the same way as the vernier on the height gage, except that the distance between the graduations on the bar of the gear tooth caliper is .020" instead of .025", as on the bar of the height gage. The vernier of the gear tooth caliper has only 20 divisions, while the height gage vernier has 25 divisions.

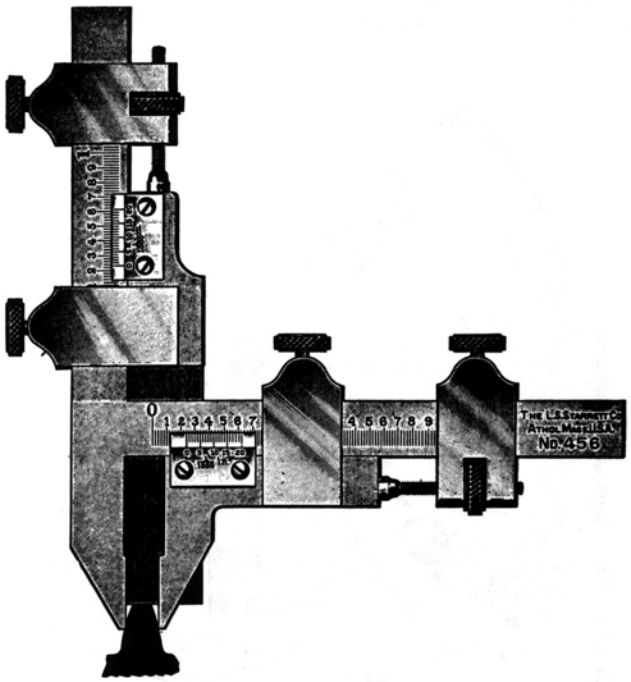


Fig. 290

4. Describe the vernier bevel protractor.

A. The vernier bevel protractor (shown in Fig. 291) is a steel tool having a dial graduated in degrees and a sliding blade which is usually about one-sixteenth of an inch thick. One side of the tool is flat, permitting it to be laid flat upon the work.

The disk of the bevel protractor is graduated in degrees throughout the entire circle. The vernier is graduated so that 12 divisions on the vernier occupy the same space as 23 degrees on the disk. The difference between the width of one of the 12 spaces on the vernier and two of the 23 spaces on the disk is therefore one-twelfth of a degree. The only difference between a vernier on a bevel protractor and a vernier on a height gage is that the vernier on the bevel protractor is graduated in degrees instead of inches.

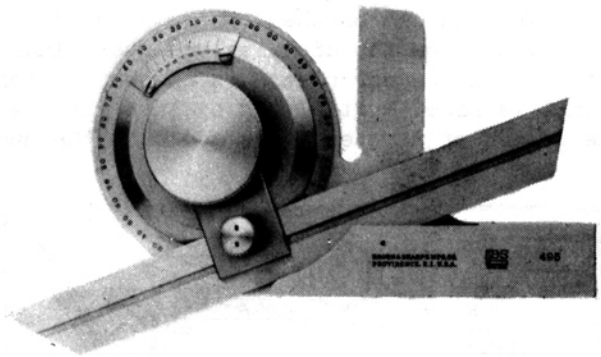


Fig. 291

First of all, observe that the vernier covers a space of 23 degrees or 1380 minutes. As it is divided into 12 equal parts (see Fig. 292), each part is equal to one-twelfth of 1380 minutes or 115 minutes, as indicated at B. Since there are 60 minutes in one degree, the space of two degrees indicated by C is equal to 120 minutes. Solving for A, C - B = A or 120 - 115 = 5 minutes, which is the curved distance the vernier moves from one line on the vernier to the next line on the protractor disk.

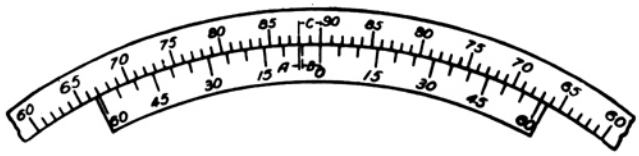


Fig. 292

To read the protractor, note on the disk the number of whole degrees between 0 on the disk and 0 on the vernier. Then count in the same direction the number of spaces from 0 on the vernier to a line coinciding with a

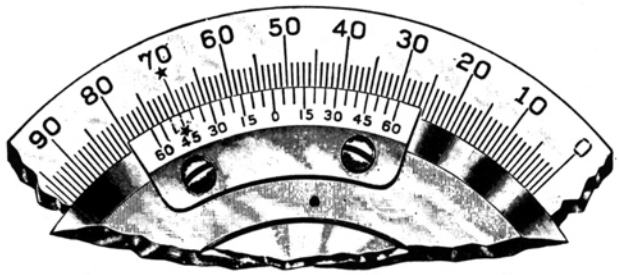


Fig. 293

line on the disk. Multiply this by five and the product will be the number of minutes to be added to the number of whole degrees. For example, in Fig. 293 the number of degrees between 0 on the disk and 0 on the vernier is 52. The line 45 on the vernier coincides with line 70 on the disk, as the stars indicate. The number of spaces on the vernier from 0 is nine. Multiplying by five gives 45, the number of minutes to be added to the number of degrees. The reading of the protractor is therefore 52 degrees and 45 minutes ($52^{\circ}45'$).

As the divisions, both on the disk and the vernier, are numbered both to the right and the left from zero, any size angle can be measured, and the readings on the disk and the vernier are taken either to the right or the left, according to the direction in which the zero on the vernier is moved.

Fig. 294 shows some of the applications of the vernier bevel protractor.

The draftsmen's protractor in Fig. 295 is graduated in degrees to read either to the right or the left, with vernier reading in spaces of five minutes. The three straight edges of the protractor are graduated in inches and sixteenths.

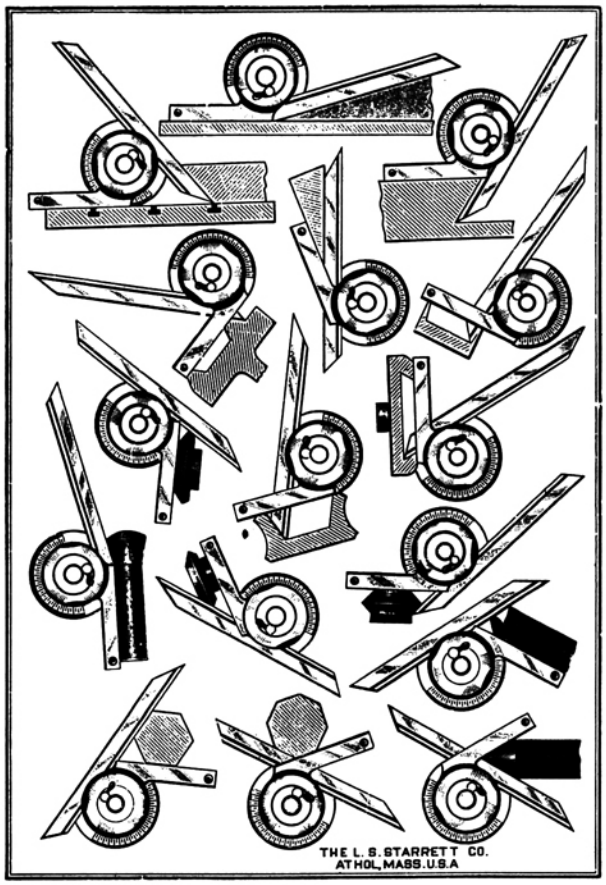


Fig. 294

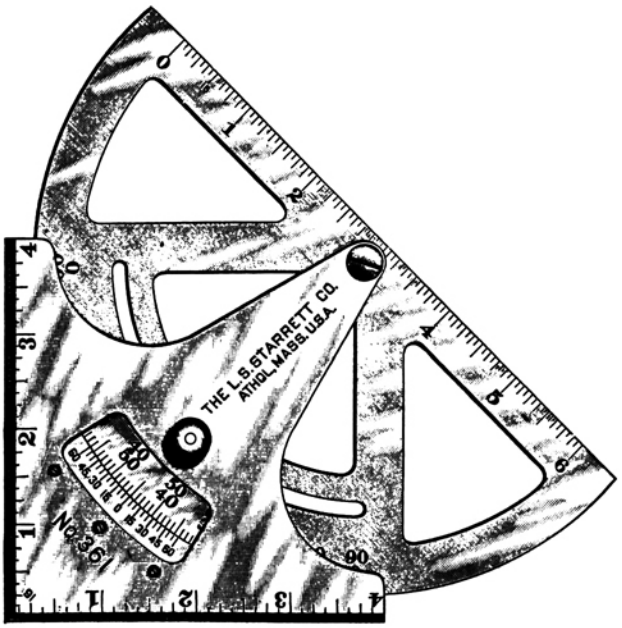


Fig. 295

Chapter 7

CHISELS AND CHIPPING

1. What is a chisel?

A. A chisel is a metal tool or instrument with a cutting edge at the end of the blade, used in dressing, shaping, or cutting. It is driven by a hammer or mallet.

2. What is chipping?

A. Chipping is the process of removing stock with a hammer and chisel.

3. Name two methods of chipping.

A. Hand and pneumatic.

4. Describe the method of chipping by hand.

A. A hammer, weighing from one to one and three-quarter pounds, and a variety of chisels are used for ordinary chipping. The chisels commonly used are flat, but the cape chisel and various forms of side and grooving chisels are also used. The hammer should be held at the extreme end when chipping, grasped by the thumb and second and third fingers, with the first and fourth fingers closed loosely around the handle. The hammer handle may thus be swung more steadily and more freely without tiring the hand as much as if the handle were grasped rigidly by all four fingers.

The chisel should be grasped with the head close to the thumb and first finger, and held firmly with the second and third fingers. The first finger and thumb should be slack, as the muscles are then relaxed and the fingers and hand are less likely to be injured if struck with the hammer. The edge of the chisel should be held on the point where the cut is desired, at an angle that will cause the cutting edge to follow the desired finished surface. After each blow the chisel must be set to the proper position for the next cut. The depth of the cut depends on the angle at which the chisel is held.

5. From what kind of material are chisels made?

A. Chisels are made from a good grade of octagon-shaped steel, generally known as chisel steel. They are forged, usually annealed, and then hardened and tempered.

6. How are chisels classified for size?

A. By the size of the cross section.

7. Why are chisels annealed?

A. Annealing makes a chisel tougher and stronger.

8. What part of a chisel is hardened?

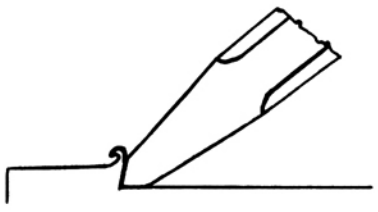


Fig. 296. Correct angle at cutting edge.

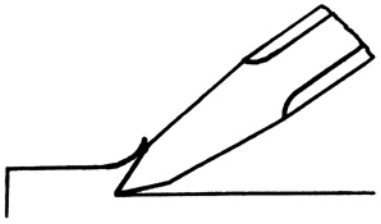


Fig. 297. Angle at cutting edge is too small for general use.

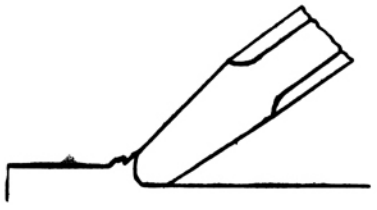


Fig. 298. Point rounded from being used after it became dull.

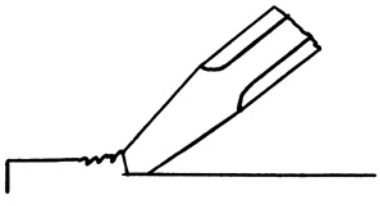


Fig. 299. Dull and incorrectly sharpened. Cutting angle should be smaller.

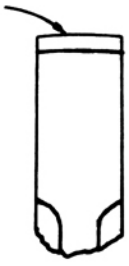


Fig. 300. Cutting edge slightly rounded to give better cutting action.

A. The cutting edge is hardened, from the end back to approximately one inch from it.

9. What angle should be ground on the cutting edge of a chisel?

A. The correct cutting angle of a chisel depends on the strength of the stock to be chiseled. The softer the metal, the sharper should be the cutting angle of the chisel. An angle of about 70° is suitable for most work. Correct cutting edges are illustrated in Figs. 296 and 300. Avoid tool edges as shown in Figs. 297, 298 and 299.

10. Draw sketches of the flat, cape, diamond point, and round point chisels, showing the "facet" on the flat chisel.

A. The appearance of each of these chisels is illustrated in Fig. 301.

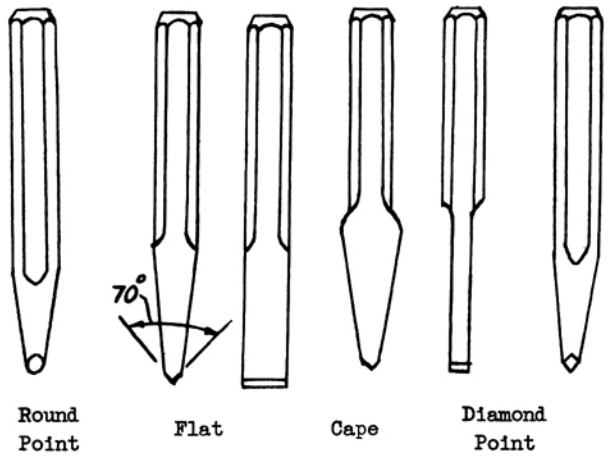


Fig. 301

11. How should a cold chisel be sharpened?

A. The chisel should be held at the correct angle and moved across the face of the wheel. The pressure of the chisel against the wheel must not be strong enough to draw the temper from the chisel. The facet should be straight back of the cutting edge, but across the width of the chisel it should be curved a little to give a slightly convex cutting edge.

12. When should the round nose chisel be used?

A. The round nose chisel is used for chipping filleted corners and concave surfaces.

13. When should the flat cold chisel be used?

A. The flat cold chisel is the best chisel for chipping flat surfaces or cutting through thin metal.

14. When should the cape chisels be used?

A. The cape chisel is a narrow chisel for chipping grooves, slots, etc. The round nose cape chisel is used on drill press work, in drawing the drill back concentric with the layout.

15. When should the diamond point chisel be used?

A. The diamond point chisel should be used in cutting V-shape grooves.

16. Give some hints on chipping.

A. Place a chipping guard (a piece of canvas about two feet square attached to two wooden pedestals) in front of the work you are chipping so that flying chips will not injure the fellow in front of you. Wear goggles to protect your eyes.

Use a packing block when chipping in a vise. Never hammer the handle of a vise. The vise jaws should have guards made of some soft material, such as brass or copper, to protect the finish on the work.

17. When chipping, should the cutting edge or the head of the chisel be watched?

A. Watch the cutting edge of the tool. With practice, the ability to hit the head of the chisel without watching it is soon acquired.

18. What is meant by a "Mushroom Head" on a chisel?

A. A Mushroom Head on a tool is a head that has been hammered until the end spreads out so as to resemble a mushroom.

19. Why are "Mushroom Head" chisels dangerous?

A. The mushroomed part of the head may break off and injure some one. The chisel cut cannot be controlled so easily when the head is mushroomed.

20. Write a 100 word story about chisels and chipping, paying particular attention to the points shown in Fig. 302.

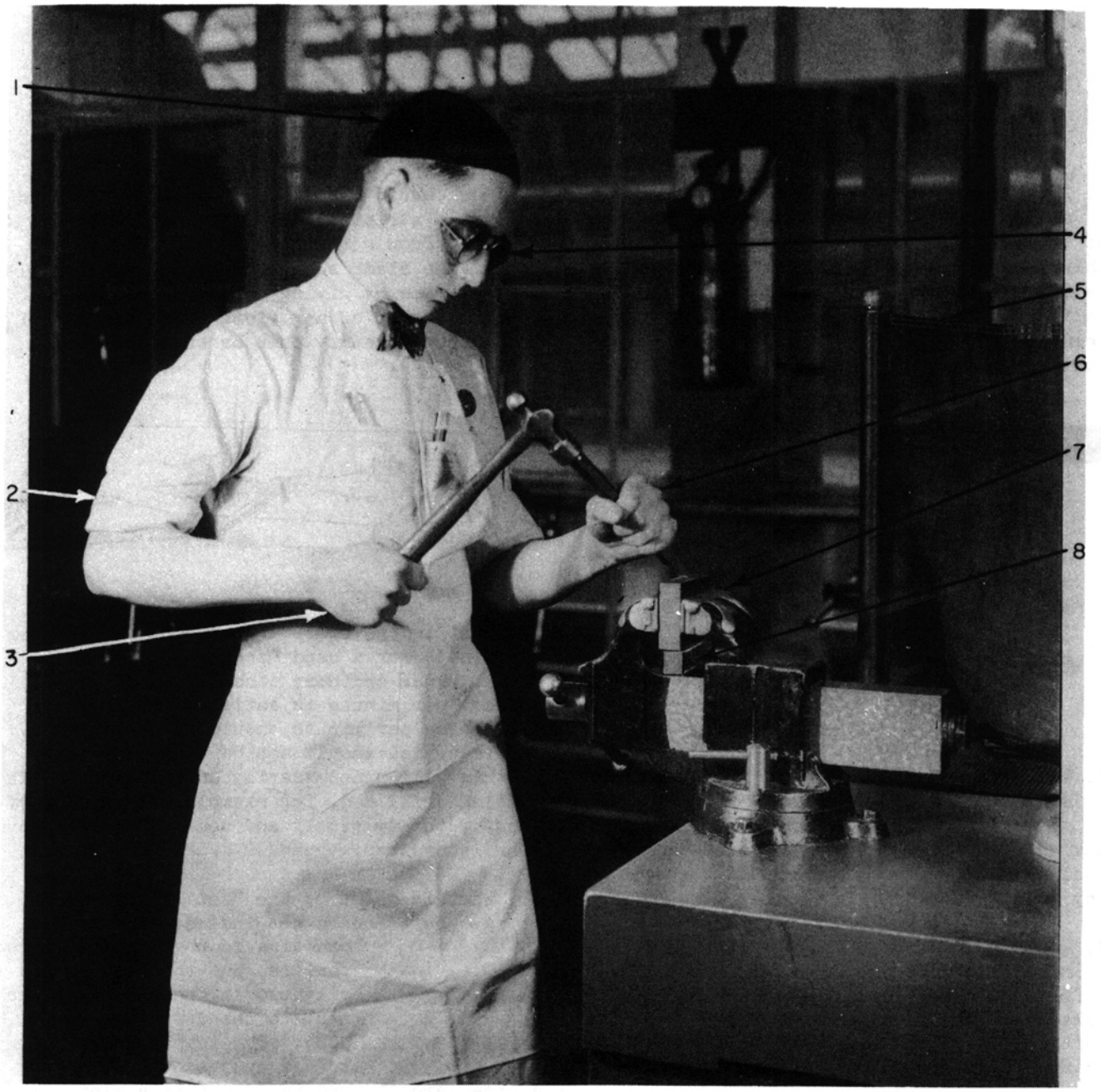


Fig. 302

Chapter 8

HACK SAWS AND SAWING

1. Describe the hack saw blade.

A. The hand hack saw blade (Fig. 303), one of the most useful tools in the shop, is a thin blade with teeth formed on one edge, and is 6 to 12 inches long, approximately one-half inch wide, and usually about .027" thick. Held in a frame (Fig. 304), it is used for cutting metal.

Fig. 303

Fig. 304

2. What kind of steel is used in making hack saw blades?

A. Different kinds of steel are used in making blades for different classes of work. Tool steel, high speed steel, or tungsten alloy steel are generally used.

3. What is meant by "flexible back" and "all hard" blades?

A. A "flexible back" blade is one in which only the teeth are hardened, leaving the back soft. When the entire blade is hardened it is called an "all hard" blade.

4. How should a hack saw blade be placed in a frame?

A. Have the teeth of the blade pointing forward so that the forward stroke will be the cutting stroke (Fig. 304). The blade should be drawn tight enough so that it will not bend. In using a flexible blade the tension should be increased while cutting, as the blade will stretch because of the heat produced by friction.

5. Does a saw blade cut on the return stroke?

A. No; because the teeth are formed so as to cut only on the forward stroke.

6. How should a blade be held in a frame to cut a strip 18 inches long and 2 inches wide from a thin sheet of metal?

A. The blade should be set at right angles to the frame by giving the clips a quarter-turn. A strip of any width up to the capacity of the frame may then be cut (see Fig. 305).

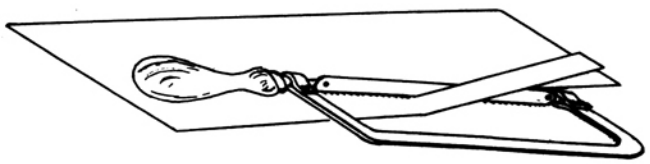


Fig. 305

7. What is the correct number of strokes per minute for a hand hack saw blade?

A. Under ordinary conditions, 35 to 40 strokes per minute is satisfactory. About 50 strokes per minute should be the maximum. Hard material should not be sawed too fast, as this will unnecessarily dull the blade. In cutting drill rod, for example, it is very effective to saw slowly and use greater pressure than is ordinarily used.

8. What is meant by the "set" of a saw?

A. The standard set indicated by general practice is regular alternate; that is, one tooth is turned slightly to the right and the next to the left (see Fig. 306). The teeth should be turned just enough to insure a free, smooth, and rapid cut in a slot a little wider than the blade itself, removing no more stock than is necessary. In certain fine-toothed saws every pair of teeth is set alternately right and left, a style of setting known as double alternate (see Fig. 307).

Fig. 306

(a) (b)

Fig. 307

9. When should an all hard saw blade be used? A flexible or soft back blade? Why?

A. Use an all hard blade on brass, tool steel, cast iron, rails, etc., and a flexible blade on channel iron, tubing, tin, copper, aluminum, babbitt, etc. An all hard blade is used on the former group of metals because it does not have the tendency to buckle or run out of line, when pressure is applied to it, that the flexible blade has. The flexible blade is used on the latter group because it does not break as easily as the all hard blade on a material with a thin cross section.

10. Why should a blade with the correct number of teeth per inch be used on a particular job?

A. The greatest economy is obtained by using blades of the correct pitch (distance from a point on one tooth to a corresponding point on the next tooth). Fig. 308 shows the importance of correct pitch.

11. On what kind of a job should a blade with 14 teeth per inch be used? Why?

A. Use a blade with 14 teeth per inch on machine steel, cold rolled steel, and structural steel. The coarse pitch makes the saw free and fast cutting (see Fig. 308).

12. When should a blade with 18 teeth per inch be used? Why?

A. Use this blade on solid stock, aluminum, babbitt, tool steel, high speed steel, cast iron, etc. This pitch is recommended for general use, as a blade with 14 teeth per inch is too coarse and leaves the cut too ragged (see Fig. 308).

13. On what kind of a job should a blade with 24 teeth per inch be used? Why?

A. Use this blade on tubing, tin, brass, copper, channel iron, and sheet metal over 18 gage. If a coarser pitch is used, the thin stock will tend to strip the teeth out of the saw blade. Two or more teeth should be in contact with the work (see Fig. 308).

14. When should a blade with 32 teeth per inch be used? Why?

A. Use this blade on small tubing, conduit, and sheet metal less than 18 gage, because these thin materials require a very fine pitch in the saw blade (see Fig. 308).

15. How much pressure is required for sawing with a hand hack saw?

A. A pressure of 20 to 30 pounds per inch of contact area of the teeth has been found to give the most satisfactory results. Enough pressure must be applied so that the saw will not slip over the work, but not enough to cause the blade to buckle or break. More pressure must be applied as the saw becomes dull.

16. What is the effect of using a coolant on a power hack saw blade?

A. By using a coolant the strokes per minute may be increased and the saw will last much longer. The coolant absorbs the heat due to friction and prevents the blade from overheating and losing its temper.

17. How should thin steel stock be supported while being cut with a hack saw?

A. Clamp thin stock between two pieces of wood or soft steel and saw through all three pieces.

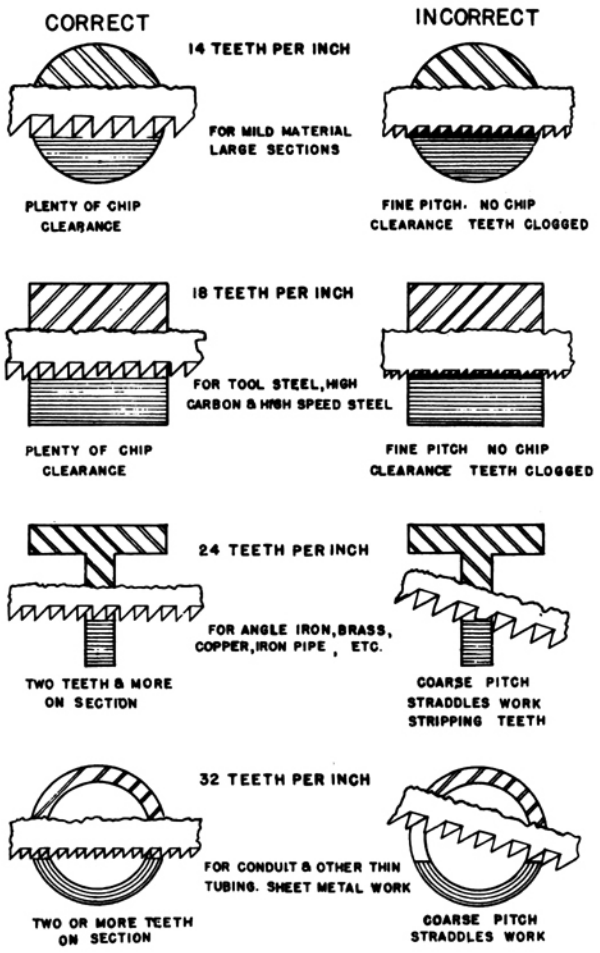

CORRECT
INCORRECT
14 TEETH PER INCH
FOR MILD MATERIAL LARGE SECTIONS
PLENTY OF CHIP CLEARANCE
FINE PITCH. NO CHIP CLEARANCE TEETH CLOGGED
18 TEETH PER INCH
FOR TOOL STEEL, HIGH CARBON & HIGH SPEED STEEL
PLENTY OF CHIP CLEARANCE
FINE PITCH NO CHIP CLEARANCE TEETH CLOGGED
24 TEETH PER INCH
FOR ANGLE IRON, BRASS, COPPER, IRON PIPE, ETC.
TWO TEETH & MORE ON SECTION
COARSE PITCH STRADDLES WORK STRIPPING TEETH
32 TEETH PER INCH
FOR CONDUIT & OTHER THIN TUBING. SHEET METAL WORK
TWO OR MORE TEETH ON SECTION
COARSE PITCH STRADDLES WORK

Fig. 308. Chart for selecting correct number of teeth per inch for hack saw blades for various jobs. In hand frames, a blade with 18 teeth per inch is recommended for general work.

18. Name three common causes for the breaking of hack saw blades.
A. (1) Using a coarse blade on thin work;
(2) Drawing the blade too tightly and then canting (tilting) it over;
(3) Using too much pressure.

19. Describe slotting hack saw blades.
A. Slotting hack saw blades (illustrated in Fig. 309) are usually eight inches long by one-half inch wide, and of four different thicknesses (about .049", .065", .083", and .109"). They are very handy when slotting a few screws for a special job which is needed at once. When slotting saw blades are not available, two ordinary saw blades may be placed in the frame side by side and used as a substitute.

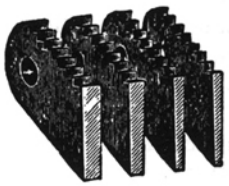


Fig. 309

20. If a saw blade does not cut straight, how can the condition be corrected?
A. First examine the blade to see that the teeth are not worn on one side, as shown in Fig. 310. If worn, replace with a new blade. If not worn, start a new cut.

Fig. 310

21. Why should a new cut be started after replacing a worn blade with a new one?
A. The set of the old blade will be worn slightly and the cut will be narrower than the new blade. The new blade will break if it is forced into the old cut.

24. Explain the importance of holding work in the vise correctly.
A. It is important that the work be placed in the vise correctly if the greatest efficiency is to be secured from the saw blade. Expose as much of the surface of the work as possible, so that a corner may be taken gradually and the maximum number of teeth engaged throughout the cutting. This prevents stripping, the cut is made quicker and straighter, and the blade lasts longer. Be sure that the work is held rigid, and always start the cut with the least possible angle facing the thrust of the saw teeth. It is especially important to consider these conditions in cutting angle iron or material with an odd shape.

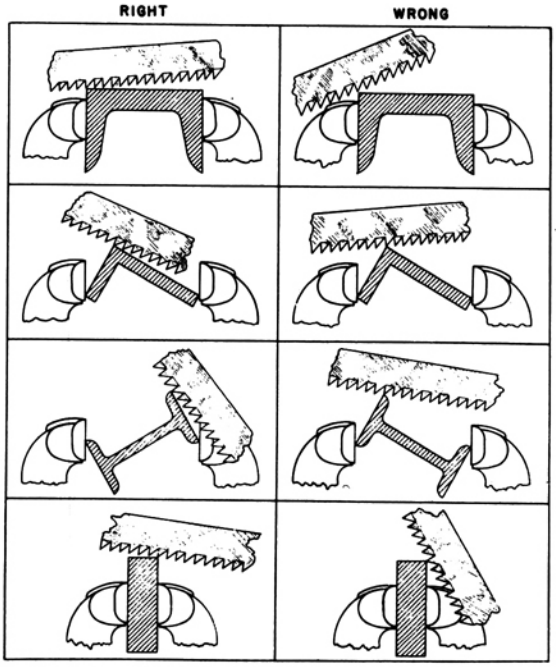


Fig. 312

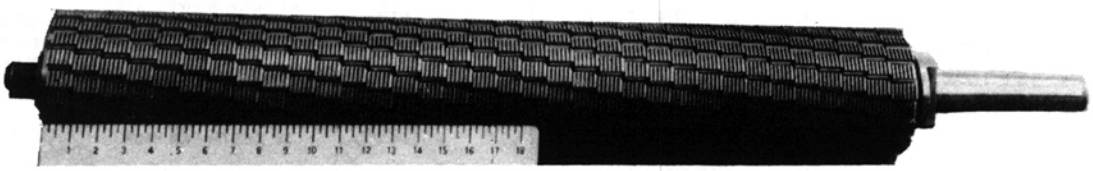


Fig. 311

22. How are teeth formed on hack saw blades?
A. The teeth are formed by using gang milling cutters (illustrated in Fig. 311).

23. How should the length of a hack saw blade be measured?
A. The length is the distance between the centers of the holes in the blade.

Fig. 312 shows the right and wrong positions of shapes commonly cut with a hack saw.

25. Name two methods, other than the hand hack saw method, that are used in sawing metal.
A. The power hack saw and the band saw are also used for sawing metal.

26. Give some of the rules to be followed in sawing with a hand hack saw.

A. (1) Use a blade with the correct pitch.
(2) Saw as close as possible to the point where the work is clamped, to prevent chattering.
(3) Do not cut too fast.
(4) Relieve pressure on the back stroke.
(5) Do not use excessive pressure.
(6) Saw carefully when the blade is almost through the cut.

Chapter 9

FILES AND FILING

1. What is a file?

A. A file is a hardened steel instrument having cutting edges or teeth upon its surface, made by rows of straight chisel cuts running diagonally across it (see Fig. 313). It differs from the chisel in having a large number of cutting points instead of one cutting edge and in being driven directly by hand instead of by a hammer. The file is used for abrading or smoothing metals.

4. What is meant by the "safe edge"?

A. The safe edge of the file is the edge on which no teeth are cut. The mill file and the flat file have single teeth cut on the edges, but the hand file usually has teeth on only one edge, the other being termed the safe edge. The pillar file has two safe edges.

5. What is meant by a "blunt" file?

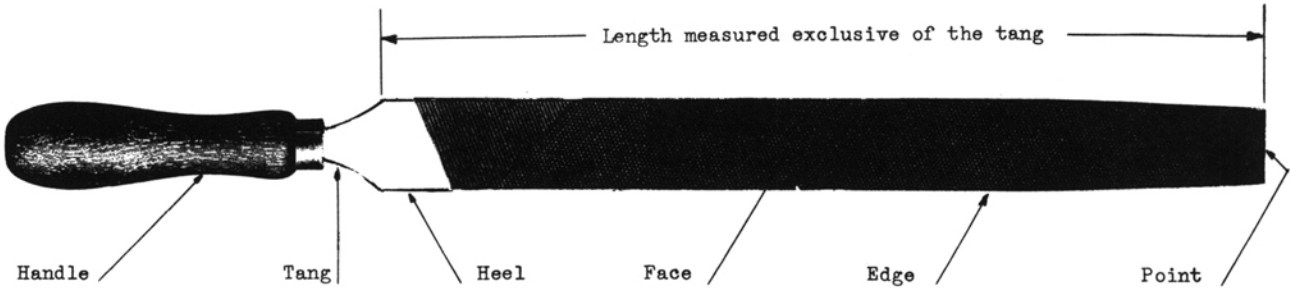


Fig. 313

2. What is a rasp?

A. The rasp, shown in Fig. 314, is similar to a file but has coarse teeth raised by a pointed triangular punch.

A. A blunt file is a file that has the same sectional shape throughout its length, from point to tang.

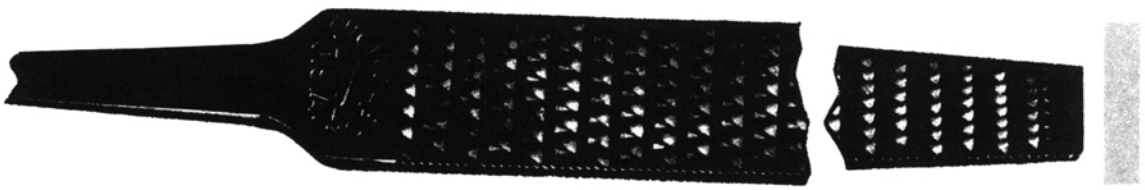


Fig. 314

3. What is the tang of a file?

A. The tang is the pointed portion of the file which is inserted in the handle. CAUTION: Be sure that the handle is firmly attached to the file.

6. Why are files made with convex surfaces?

A. Files are generally made with convex surfaces, that is, they are thicker in the middle than at the ends (see Fig. 315). This is done to prevent all the teeth from cutting

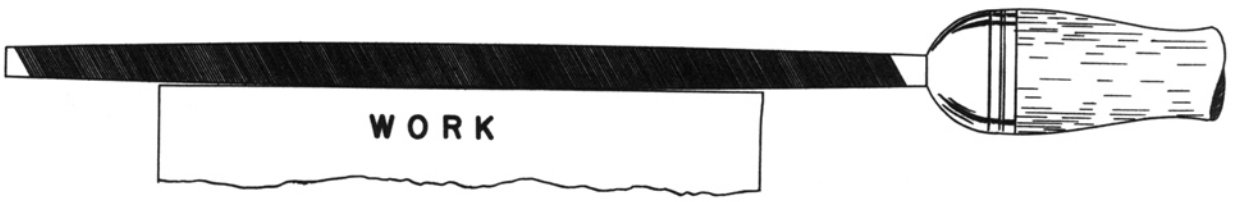


Fig. 315

at the same time, as that would require too much pressure on a file and make it hard to control. A flat surface could not be obtained if the face of the file were straight, as there is a tendency to "rock" the file. The convex surface helps to overcome the results of rocking.

This convexity of files also serves another purpose. The pressure applied to a file to make it bite into the work also bends the file more or less, and if the file in its natural state were perfectly flat, it would be concave during the cutting operation. This would prevent the production of a flat surface, as the file would cut away at the edges of the work and leave a convex surface.

7. What is meant by the "taper" of files?

A. Files taper slightly, being slightly smaller in width toward the point (see Fig. 316). This does not apply to blunt files.

Fig. 316

8. How are files classified?

A. They may be divided into two classes, single cut (Fig. 317) and double cut (see Fig. 318).

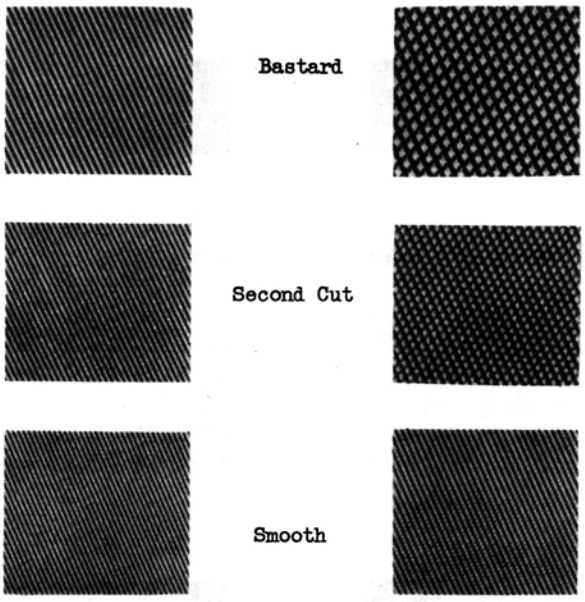


Fig. 317. Types of Single Cut Files

Fig. 318. Types of Double Cut Files

9. Describe the use of the following files: single cut (Fig. 317), double cut for general work (Fig. 318), and double cut for finishing work (Fig. 319).

A. The single cut mill files of Fig. 317 derive their name from the fact that they were first used for filing mill saws. They are also used for lathe filing, draw filing, and for finishing various compositions of brass and bronze. This type of file leaves a fine finish. It differs from other files in having single cut teeth.

The flat double cut files shown in Fig. 318 are used by machinists, machinery builders, ship and engine builders, repair men, and tool makers when a fast cutting file is needed. This type of file leaves a rough finish.

The hand finishing double cut file shown in Fig. 319 is used by machinists and tool makers when a smooth finish is desired.

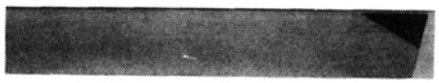


Fig. 319. Double Cut Finish File

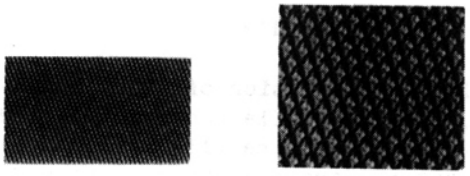
Double cut files are fast cutting but cannot be used for draw filing. The finish produced by a double cut file is not as smooth as that produced by a single cut file.

10. What is the meaning of such terms as rough, coarse, bastard, second cut, smooth, and dead smooth?

A. These terms refer to the distance apart of the parallel cuts. A numbering system is used for the same purpose on small files, ranging from 00 to 8, 00 being the coarsest. Bastard, second cut, and smooth files are illustrated in Figs. 317 and 318.

11. Is the pitch of a 16" second cut file the same as the pitch of a 6" second cut file?

A. No, the shorter the file the finer the pitch (as the illustrations in Fig. 320 show clearly).



6" 16"

Fig. 320

12. How is the length of a file measured?
A. From the point to the heel (see Fig. 313).

13. What are some of the different shapes of files?
A. Cross sections of some of the most commonly used files are shown in Fig. 321.

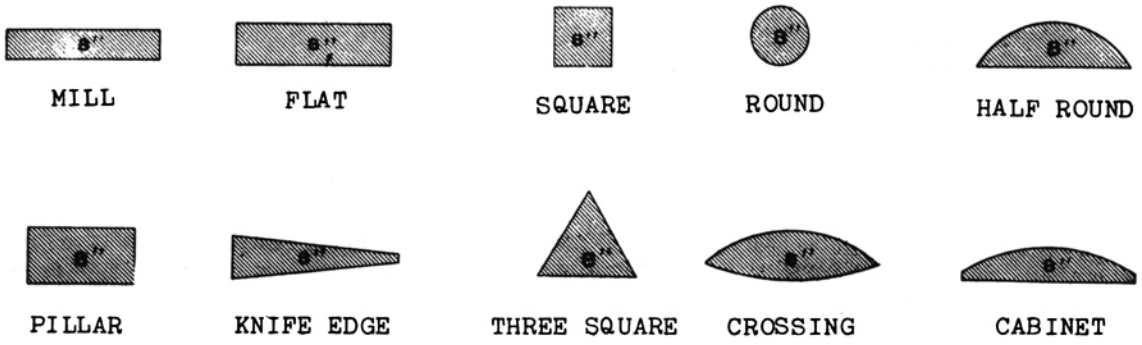


Fig. 321

14. Describe the "pillar" file.
A. The pillar file shown in Fig. 322 is similar to the hand file shown in Fig. 318 except that it is narrower and one or both edges are safe edges (have no teeth cut on them). The pillar file is used for filing slots and keyways and for filing against shoulders.

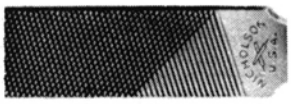


Fig. 322

15. Give a description of the "square" file.
A. The square file is similar to the mill file with the exception that its cross section is square and it has double cut teeth on all four sides (see Fig. 323). It is used for filing small square or rectangular holes, for finishing the bottoms of narrow slots, etc.

Fig. 323

16. Give a description of the "round" file.
A. The round file shown in Fig. 324 is similar to the square file with the exception that the cross section is round. It is generally tapered and the small sizes are often called "rat-tail" files. It is used for enlarging round holes and for finishing fillets.

Fig. 324

17. Describe the "three-square" file.
A. The three-square file shown in Fig. 325 (commonly called the three-cornered file) is triangular in section, with angles of 60°. It tapers to the point, the corners are left sharp, and it is double cut on three sides and single cut on the edges. It is generally used for filing internal angles more acute than those of the rectangle, for clearing out square corners, and for filing taps, cutters, etc.

Fig. 325

18. Give a description of the "half-round" file.
A. Fig. 326 shows the cross section of the "half-round" file (observe that it is not a half circle). It derives its name from the fact that one side is rounded while the other is flat. It can be used for various jobs on which no other type of file would be satisfactory.

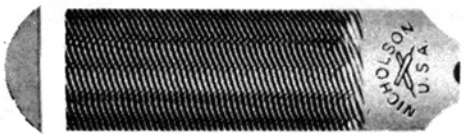


Fig. 326

19. Describe the "crossing" file.

A. The crossing file shown in Fig. 327 has a double circular section, one side having the same radius as the half-round file and the other side having a flatter curve, or larger radius. It tapers to the point in both width and thickness and is double cut on both sides. This file is often used in place of the half-round file.

22. When should the coarser files be used?

A. A coarse file should be used only when a comparatively large amount of material is to be removed.

23. When should the finer files be used?

A. A fine file should be used only for finishing work.

Fig. 327

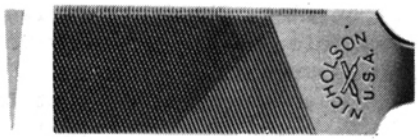


Fig. 328

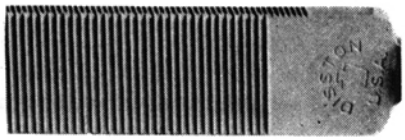


Fig. 329

20. Describe the "knife" file.

A. The knife file shown in Fig. 328 is made from steel that is knife shaped, the included angle of the sharp edge being approximately 10°. This file tapers to the point in width and thickness, and is double cut on both flat sides and single cut on both edges. It is used for finishing the sharp corners of many kinds of slots and grooves.

21. What kind of a file should be used for filing lead or babbitt?

A. Use either the "lead float" file shown in Fig. 329 or the curved tooth "Vixen" file shown in Figs. 330 and 331.

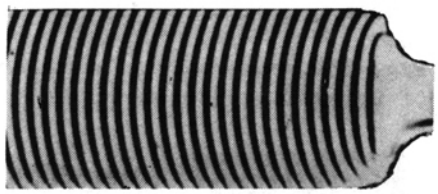


Fig. 330

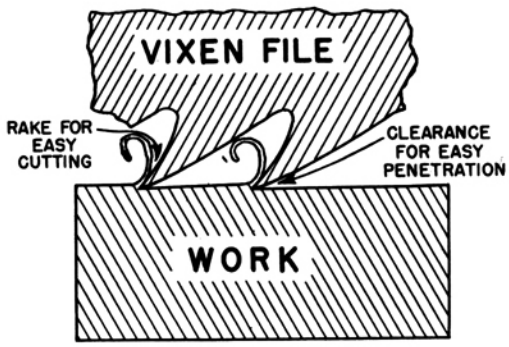


Fig. 331

24. What kind of a 12"-file should be used to remove stock rapidly?

A. A double cut bastard or double-cut coarse-tooth file should be used to remove stock rapidly.

25. What file should be used for finishing?

A. A single cut fine tooth file should be used for finishing.

26. What is the proper way to hold a file?

A. As illustrated in Fig. 332, grasp the handle in the right hand so that it rests against the palm of the hand, with the thumb placed on top. Place the left hand at the end of the file and let the fingers curl under it.

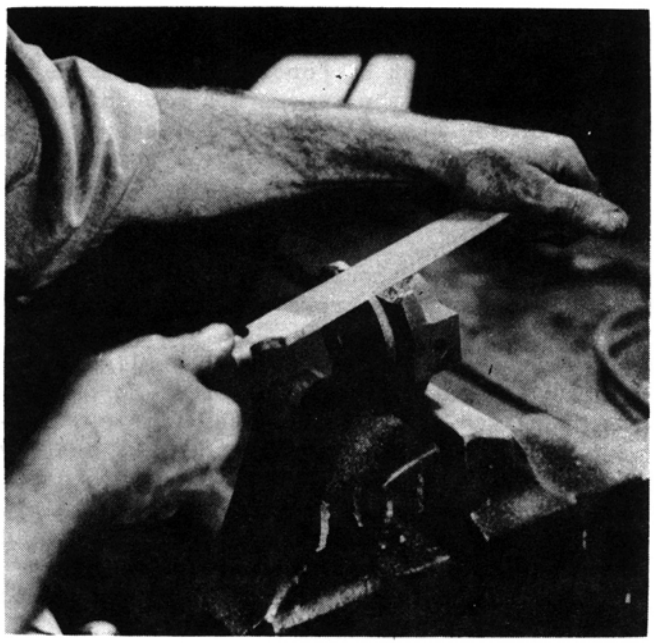


Fig. 332

27. What should be the position of the body when filing?

A. It is important to have the body in the correct position, as the muscles must move freely. The left foot should point forward and the right foot be brought up close enough to the left to give the necessary balance. When filing, the body should lean forward on the forward stroke for part of the stroke and then return to the original position at the finish. The file must be held straight or the surface will not be flat. The strokes should not be too fast as this will ruin the file and the work. Enough pressure should be applied to make the file cut evenly.

28. What is meant by "draw filing"?

A. With the file in a position similar to that shown in Fig. 333, grasp it with both hands, the thumbs being one-half or three-quarters of an inch from each side of the work, and pull or push the file. By using a single cut file in this manner, the teeth will have a shearing action and give a smooth finish to the work.

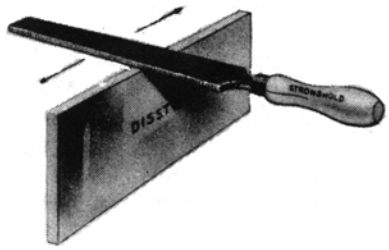


Fig. 333

29. What is meant by "crossing the stroke"?

A. Crossing the stroke means changing the angle of the stroke to about 45 degrees from the previous strokes. This will show the high spots and also tends to keep the work flat.

30. Should the file be lifted from the work on the return stroke? Explain thoroughly.

A. Be sure to raise the file slightly during the return stroke in order to clear the work and avoid dulling the file by wearing away the back of the teeth, thus destroying the cutting edges. This procedure does not hold true in the filing of soft metals, such as lead or aluminum, as the file should be drawn back along those metals on the return stroke, as an aid in cleaning the teeth.

31. What precaution should be taken before filing a cast iron casting?

A. Before attempting to file a cast iron casting, the scale must be removed from its surface. This can be done by chipping, using the edge of the file, tumbling, sand blasting, or pickling (a good pickling solution is 4 to 10 parts of water to 1 part of sulfuric acid).

32. How should a piece of filed work be tested to see if it is square?

A. Two surfaces may be tested for squareness by using a solid square.

33. What three factors affect the cutting efficiency of a file?

A. The shape, sharpness, and hardness of the teeth are the important factors.

34. What is meant by "pinning"?

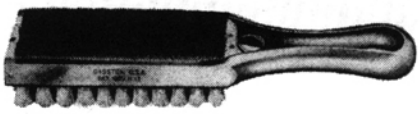
A. When filing soft metals, narrow surfaces, or corners, small particles of the material being filed tend to become clogged in the gullets between the teeth of the file, causing scratches on the work. This is called pinning.

35. What is the cause of pinning?

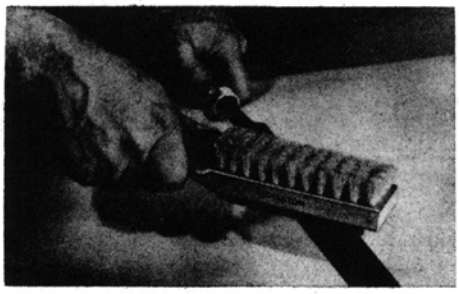
A. If the pressure applied to a file is too great, the chisel cuts on the file will clog, especially on fine cut files. In using a new file see that the rough edges and burrs are worn down slightly before taking a heavy cut. Rubbing chalk on a file will help to prevent pinning.

36. How would you clean a file that has become pinned?

A. Fig. 334 shows a file brush and the method of using it to clean a file. The file



(a)



(b)

Fig. 334. (a) File Brush, (b) Carding a File

brush is made of a rectangular piece of wood with bristles on one side and fine wires on the other. The card wire is fine enough to enter a fine tooth file and loosen chips and dirt while the brush is used to clean these out. The "scorer" in the handle of the file brush is used to remove burrs that cannot be loosened by the card. If a scorer cannot be obtained, a piece of soft metal can be used to remove material fastened in the gullets.

37. What are "increment cut" files?
A. Files were originally made by hand and the teeth were more or less irregular, according to the skill of the mechanic who cut them. Some of the teeth were higher than others, giving fewer contact points. This proved desirable as it increased the cutting life of the file, but the cost of the handmade files was necessarily great. Machines are now used to increase or decrease the cuts of the teeth, giving the same type of file as the earlier hand cut files. These are called increment cut files.

38. When does a file cut best?
A. After it has cut about 2500 strokes, or after it has removed about one cubic inch of material because at that time most of the cutting edges will be in contact with the work. It must be remembered however after continued use the worn down edges will continue to cut less and less until the life of the file is destroyed.

39. What two files are generally used in filing on the lathe?
A. The double cut flat file is used for rough filing and the single cut file is used for finish filing.

40. Why is the mill file best for filing on the lathe?
A. A mill file gives a better finish, as it is a single cut file with chisel cuts in only one direction, giving a shearing action.

41. What procedure should be used in filing on the lathe?
A. Use long, slow cuts. Be sure to bear down hard enough on the file to make it cut, but not hard enough to make it pin.

42. What is the result when short, quick strokes are taken to remove a small amount of material from a job by filing it on the lathe?
A. Either a series of small flats will be found on the periphery or the work will be out of round.

43. How much material should be left for finishing when filing on the lathe?
A. In general practice. not more than .003" is usually left for finish filing.

44. What is the effect of too much filing on the lathe?
A. The work will have a tendency to be out of round if too much stock is removed with a file.

45. What two files are most commonly used in the machine shop?
A. The flat file and the half-round file are most commonly used.

46. In placing an order for files, how should a required file be designated?
A. A file should be designated by the size, by the shape or cross section of the steel on which the teeth are cut, and by the spacing of the teeth (the size always refers to the length, which is measured from the point to the heel, and does not include the tang).

EXAMPLE: 12 - 6" Half-Round Mill Files

47. Is it ever permissible to use a file without a handle?
A. Never use a file without a handle. This is a safety rule. To do so is especially bad practice on a lathe.

48. Give two precautions that should never be overlooked when filing.
A. Never rub your hand over the work you are filing. Always make sure that filed surfaces are protected by soft material between the work and the hardened jaws of the vise.

49. Describe the use of the filing machine.
A. The filing machine shown in Fig. 335 saves a great deal of time and labor. Care must be used in setting the table to the correct clearance angle for dies, etc. Use the amount of force in setting the protractor against the file, to get the angle between the table and the file, that you would use in forcing the work against the file. The angle will vary with the amount of force used.

Fig. 336 shows the type of files used in the filing machine. In placing a file in the filing machine, the roller guide must be adjusted to give the proper amount of friction against the file. Failure to install the file correctly will cause it to break in the socket.

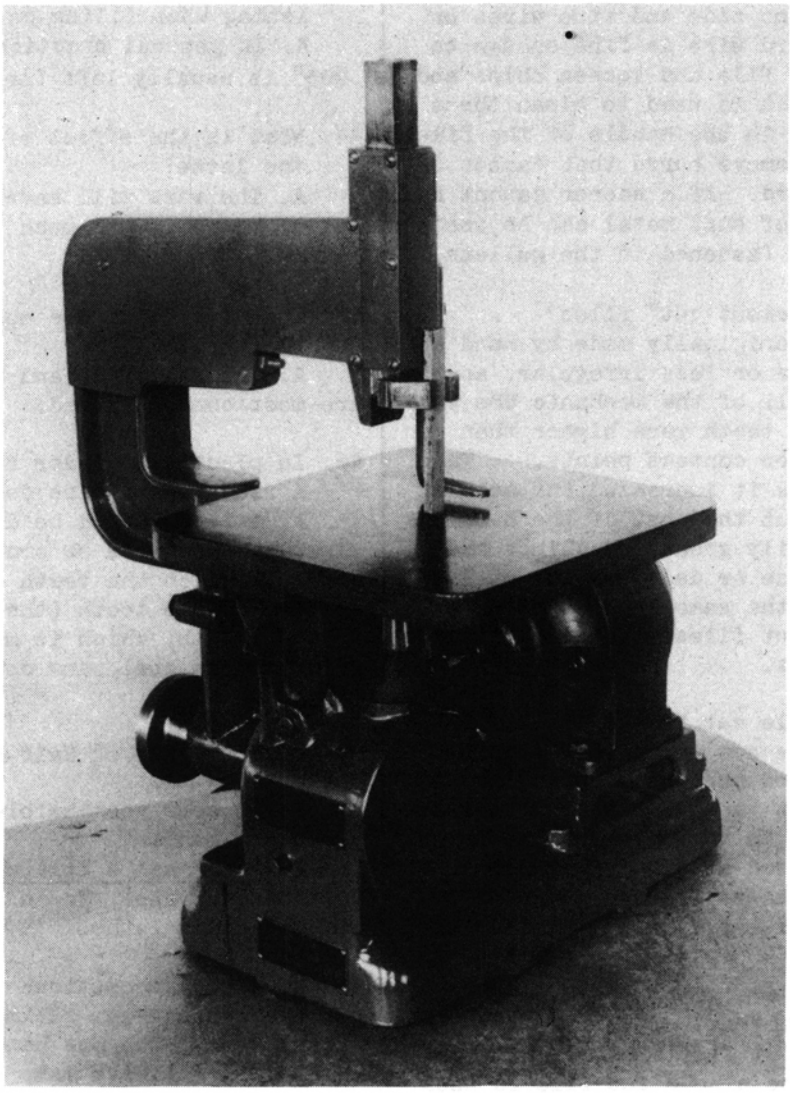


Fig. 335

Fig. 336

Chapter 10

SOLDERING

1. What is soldering?
   A. Soldering is the process of joining two metals by a third soft metal that is applied in the molten state.

2. What is the composition of solder?
   A. Solder consists of tin and lead. Bismuth and cadmium are frequently included to lower the melting point.

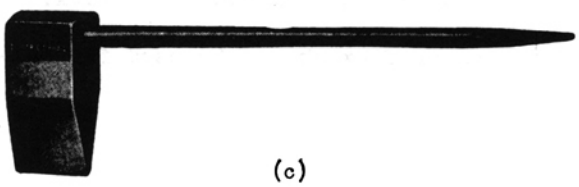
3. What is one of the most important operations in soldering?
   A. One of the most important operations, and one that is often overlooked, is that of cleaning the surface to be soldered.

4. Tell what flux is and explain why it is used.
   A. Flux is a cleanser and is used to remove and prevent the oxidation of the metals, allowing the solder to flow freely and unite more firmly with the surfaces to be joined.

5. What is a soldering copper? Make a sketch showing three different types.
   A. The soldering copper is a piece of copper attached to a steel rod with a handle. Soldering coppers are made in different lengths, forms, and weights. (See Fig. 337)

(a)

(b)



(c)

Fig. 337. (a) Square Point Copper, (b) Bottoming Copper, (c) Hatchet Copper.

6. Describe how to "tin" a soldering copper.
   A. Use a file to clean the copper back to the end of the bevel tip and heat the copper a little higher than the melting point of the solder. The way to tin the point is to rub the clean, heated copper with sal ammoniac and then apply the solder. To have a good clean point, rub over it with a rag immediately after it has been tinned.

7. How should a surface be prepared for soldering?
   A. Use a file, scraper, or acid cleaner to clean the surface.

8. Why can we not solder two pieces of metal together without the aid of a flux after their surfaces have been cleaned?
   A. A cleaned metal surface oxidizes immediately upon exposure to the air. A thin coating of oxide is formed when the oxygen and metal combine. Solder will not unite with a metal that has a coating of oxide, grease, dirt, etc. A flux is used to remove the oxide the instant the solder comes in contact with the metal.

9. What is the most commonly used flux in the machine shop?
   A. Prepared soldering paste is the most commonly used flux in the shop, while tinsmiths favor "killed" muriatic acid (zinc chloride).

10. Describe "killed" muriatic acid.
    A. Muriatic (hydrochloric) acid is killed by adding small scraps of zinc, a few at a time, until the acid fails to eat the zinc. It is then called a saturated solution. The killed or cut acid is known as chloride of zinc.

CAUTION. Do not kill the acid near any machines or tools, as the escaping fumes may cause the machines or tools to rust.

11. On what kind of metals is chloride of zinc used as a flux?

A. Chloride of zinc is used as a flux on steel, cast iron, brass, zinc, nickel, monel metal, stainless steel, lead, tin, and galvanized iron.

12. For what metal is chloride of zinc flux diluted?

A. When chloride of zinc is used as a flux for soldering tin, it is diluted with about 50% alcohol.

13. Is muriatic acid ever used in the raw state as a flux?

A. Muriatic acid is frequently used in the raw state as a flux for soldering galvanized iron. It is used in the raw state as a cleaner on cast iron, galvanized iron, and sheet steel.

CAUTION. Where muriatic acid or zinc chloride is used as a flux, the part should be cleaned after soldering to prevent the acid from eating the metal.

14. What kind of flux is used for soldering copper and brass?

A. Zinc chloride or a prepared soldering flux.

15. What kind of flux is used for soldering lead?

A. Rosin, tallow, or zinc chloride may be used as a flux for soldering lead.

16. What kind of flux is used for soldering sheet tin?

A. Beeswax, rosin, or any of the prepared fats, pastes, or liquids are good. Zinc chloride can be used by diluting it with 50% alcohol.

17. Tell how to solder cast iron.

A. Cast iron may be soldered by first scraping it, using raw muriatic acid to clean the part, and then tinning the surface to be soldered by using zinc chloride as a flux.

18. What kind of flux should be used for soldering wrought iron or steel?

A. Zinc chloride is the best flux to use for soldering wrought iron or steel.

19. Tell how to solder aluminum.

A. Flux is molded in the special solder used for aluminum. This solder requires a greater heat to melt it than the lead-tin solder. Another way to solder aluminum is to use stearin as a flux with a solder made of 70% tin, 25% zinc, 3% aluminum, and 2% tin. Paraffin or vaseline may also be used as a flux.

20. What is meant by "sweating" parts together, and why do we use this operation?

A. If the parts to be joined are perfectly tinned, flux applied between them, and then held together and heated, the parts will become perfectly soldered. This is "sweating." Split bushings are sweated together so they can be machined.

21. How can tin foil be used in soldering?

A. Tin foil is used in the operation of sweating parts together. The parts are cleaned, flux is applied, a sheet of tin foil is placed between the surfaces, and they are clamped together. Enough heat is then applied to fuse the tin foil.

22. Should a soft or hard solder be used to make a strong joint?

A. Use a hard solder, for the higher the melting point the stronger the solder.

23. What materials are used in making hard solder?

A. Hard solder, usually called spelter, is made of 4 parts of copper to 1 part of zinc. Borax is used as a flux.

24. What materials are used in making soft solder?

A. Soft solder is made of $1\frac{1}{2}$ parts of tin and 1 part of lead. This melts at a temperature of 334°F.

25. What flux is used by canneries and packing houses because of its non-poisonous qualities?

A. Rosin is generally used as a flux in the sealing of foods because it is non-poisonous. Palm oil or cocoa oil is also used.

26. What kind of flux is best to use for commutator wires and electrical connections?

A. Use an alcoholic solution of rosin for commutator wires and electrical connections. Never use an acid flux, as this will afterwards cause a corrosive action.

27. How would you solder a horizontal piece of work if the solder had a tendency to run?

A. In soldering a horizontal job, such as a piece of pipe, tin it in the regular way and make a small cup of clay to hold the solder. Clay is also used for holding small parts to be soldered.

28. What is the difference between soldering and brazing?

A. Brazing requires a greater heat and copper is used in the hard solder or spelter. A torch is generally used to heat the spelter and the part, instead of a copper as in soldering. A brazed joint is much stronger than a soldered joint.

29. How does the addition of bismuth or cadmium affect solder?

A. The addition of a small amount of either bismuth or cadmium will lower the melting point of solder.

30. What peculiar quality does bismuth have?

A. Bismuth expands when it cools.

31. What is the effect of mixing zinc with solder?

A. The addition of zinc causes solder to flow sluggishly. Aluminum has the same effect.

32. Why does solder contain lead?

A. Lead is used because it cheapens the mixture and because it has a low melting point of 620.6°F.

33. What is the effect of phosphorus on a mixture of solder?

A. The addition of a small percentage of phosphorus causes solder to flow more freely. It should be added in the form of phosphor-tin.

34. What is the correct design of the top of the square point soldering copper and how is it used properly?

A. Whatever the type of copper selected, it must have adequate capacity for the work it is to do. A perfectly soldered connection can be obtained only when the surfaces to be joined have absorbed enough heat to melt the solder. For example, it is almost impossible to solder a large vessel with a small copper, because the large vessel will absorb all the heat from the small copper and the part to be soldered will not be heated sufficiently to cause a good fusion. As a large copper will carry more heat to the part being soldered, a large copper should be used on large jobs. A small copper should of course be used for small or intricate jobs.

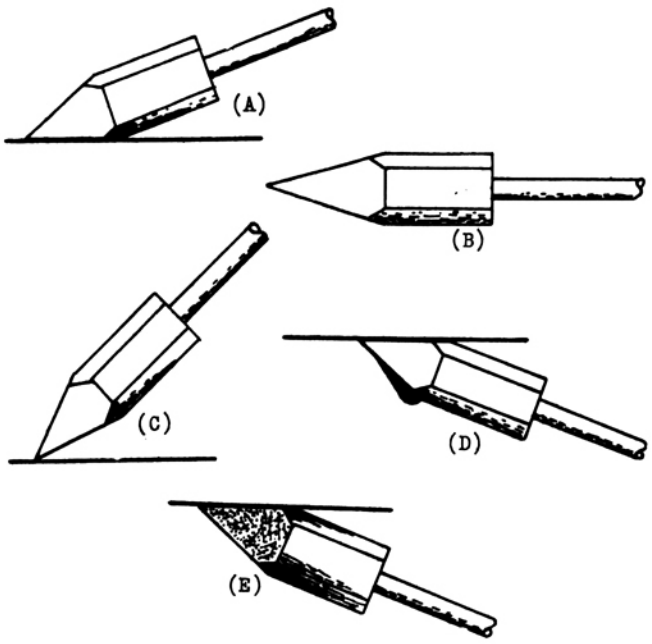


Fig. 338

The size of the copper is important, but it must also have the proper shape and be used efficiently. At A in Fig. 338, the heat can be transmitted as rapidly as possible. The copper at B is too pointed and the heat cannot be delivered properly. The thick and heavy-pointed copper at C is shaped properly but it is being applied at the wrong angle to pass the heat to the work. As solder will not flow upward, an attempt to solder the underside of a job by the method shown at D simply causes the solder to flow away from the joint. However, the following method can be used to solder this job. Clean only one side of the soldering copper, heat it, tin it, and then apply it to the work. Solder can be applied in this way because it will cling only to the clean side of the copper. Be sure that the dirty side shown at E and its opposite side are left dirty so that the solder will not run off.

Therefore the proper design and application of the soldering copper is shown at A, the point of the copper at B being too thin and the two coppers at C and D being held in wrong positions.

Chapter 11

SHOP REVIEW

1. What is an inch?

A. An inch is a standard unit of length or measurement. 12 inches = 1 foot, 3 feet = 1 yard, $16\frac{1}{2}$ feet = 1 rod, 5280 feet = 1 mile, 160 square rods = 1 acre.

2. What parts of an inch are generally used for measurements in the shop?

A. The usual units are halves, quarters, eighths, sixteenths, thirty-seconds, sixty-fourths, and thousandths. Ford Motor Company has adopted a decimal system for specifying divisions of an inch on new designs using tenths, hundredths, and thousandths of an inch as units of measurements. The rule shown in Fig. 339 is graduated into tenths and fiftieths of an inch.

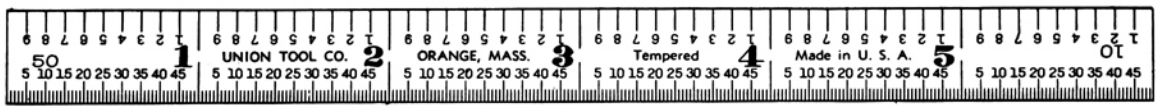


Fig. 339

3. How many eighths are there in an inch? In $\frac{1}{4}$"? In $\frac{1}{2}$"? In $\frac{3}{4}$"?

A. $1" \times \frac{8}{8} = \frac{8}{8}$; $\frac{1}{4}" \times \frac{2}{2} = \frac{2}{8}$; $\frac{1}{2}" \times \frac{4}{4} = \frac{4}{8}$; $\frac{3}{4}" \times \frac{2}{2} = \frac{6}{8}$.

4. How many eighths are there in $\frac{1}{2}" + \frac{1}{8}"$?

How many sixteenths in $\frac{1}{2}" + \frac{1}{4}"$?

A. $\frac{1}{2}" \times \frac{4}{4} = \frac{4}{8}$; $\frac{4}{8} + \frac{1}{8} = \frac{5}{8}$.

A. $\frac{1}{2}" \times \frac{8}{8} = \frac{8}{16}$; $\frac{1}{4}" \times \frac{4}{4} = \frac{4}{16}$; $\frac{8}{16} + \frac{4}{16} = \frac{12}{16}$.

5. How can a common fraction be changed to a decimal fraction?

A. A common fraction can be changed to a decimal fraction by dividing the numerator by the denominator. Example: 3/7 = 3 ÷ 7 = .42857.

6. How are thousandths written in the shop and on mechanical drawings?

A. Thousandths are the unit of measurement used in the shop and drawing room and are written in the decimal form. For example, 1/2 = .500; 3/4 = .750; 13/16 = .8125.

7. Write the value of .6569 in words.

A. The value of .6569 is six hundred fifty-six and nine tenths thousandths.

8. How can a fraction be reduced to its lowest terms?

A. To reduce a fraction to its lowest terms divide both the numerator and the denominator by their greatest common divisor. The greatest common divisor (G.C.D.) of two or more numbers is the largest number that will exactly divide each of them. One method used to find the G.C.D. for a fraction is as follows: Divide the larger term by the smaller term and find the remainder; divide the original divisor by the remainder; continue this process of dividing the remainder into the previous divisor until the remainder becomes zero. The last divisor is the G.C.D. If this number is one (1) the fraction cannot be reduced. This method can be applied to both proper and improper fractions.

Example: Reduce $\frac{6856}{20000}$ to its lowest terms.

6856	20000	2
6288	13712	1
568	6288	11
560	6248	14
8	40	5
	40	
	0	

Since 8 is the last divisor, it is the greatest common divisor.

$$\frac{6856 \div 8}{20000 \div 8} = \frac{857}{2500}$$

9. What is meant by the least common denominator of a group of fractions?

A. The least common denominator (L.C.D.) of a group of fractions is the least or smallest number which contains each of the denominators a whole number of times. Many times the least common denominator can be determined by inspection. When this is not possible it

can be found as follows:

Example: Find the L.C.D. of $\frac{1}{4}, \frac{2}{3}, \frac{5}{9}, \frac{3}{16}$.

First place the denominators in a row, separating them by commas, as shown at the right. Select the smallest prime number (a number that cannot be divided except by itself and one) that will exactly divide two or more of the denominators. In this case 2 will exactly divide 4 and 16. Write the quotients below, and bring down any numbers which are not divisible by this divisor. Proceed as before, again using the smallest prime number that will divide two or more of the numbers just obtained. Continue this process until no two of the remaining numbers can be divided by any number except 1. The product of all the divisors and all the numbers left in the last line of quotients is L.C.D. = 2 x 2 x 3 x 3 x 4 = 144, the L.C.D.

2	4, 3, 9, 16
2	2, 3, 9, 8
3	1, 3, 9, 4
	1, 1, 3, 4

10. Find the sum of the following:

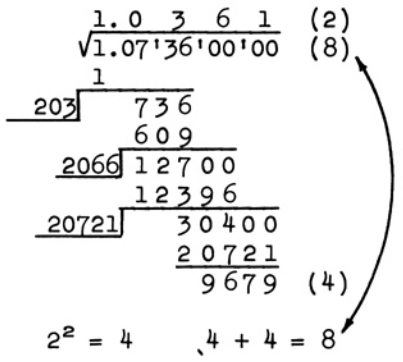
$1\frac{1}{2} + 2\frac{3}{4} + 1\frac{7}{8} + 3\frac{9}{16} + 1\frac{7}{32} + 2\frac{15}{64}$.

A. The least common denominator of all the fractions is 64; therefore, all the fractions must be changed to sixty-fourths before they can be added.

$$1\frac{32}{64} + 2\frac{48}{64} + 1\frac{56}{64} + 3\frac{36}{64} + 1\frac{14}{64} + 2\frac{15}{64} = 10\frac{201}{64}$$

$$= 13\frac{9}{64}.$$

11. Extract the square root of 1.0736 and of 125.38. Check the results.



To check: Find the excess of nines in the root. This number is squared and added to the excess of nines in the remainder. Then find the excess of nines in this sum. The result should equal the excess of nines in the original number.

12. How can the square root of a fraction be found?

A. The square root of a fraction may be found either by extracting the square root of the numerator and denominator or by changing the fraction to a decimal and extracting the square root of the decimal.

Example: $\sqrt{\frac{5}{8}} = \frac{\sqrt{5}}{\sqrt{8}} = \frac{2.2361}{2.8284} = .7905$

$$\sqrt{\frac{5}{8}} = \sqrt{.625} = .7905.$$

13. What is a circle?

A. A circle is a closed plane bounded by a curved line, all points of which are equidistant from a fixed point called the center.

14. Draw a sketch of a circle showing the different parts.

A. The sketch (Fig. 340) shows a circle

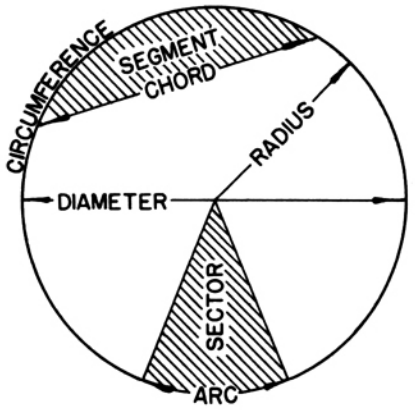
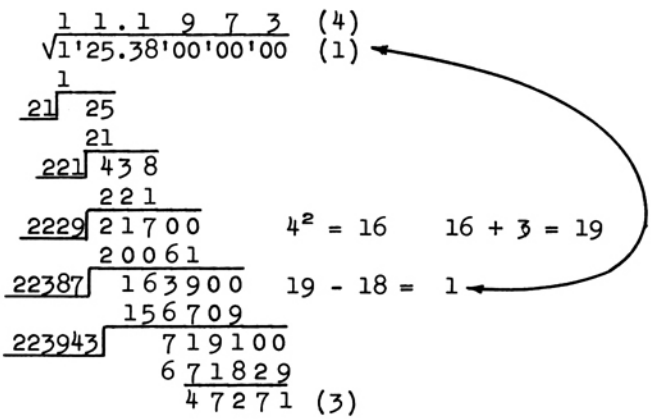


Fig. 340



with its different parts. π(pi) = 3.1416, Circumference = 3.1416 x Diameter, Radius = Diameter ÷ 2, and Area = 3.1416 x Radius squared.

15. What is meant by the circumference of a circle?

A. The circumference is the distance around a circle. It is equal to 3.1416 (π) times the diameter.

16. What is meant by the diameter of a circle?

A. The diameter is the distance across the circle, measured through the center.

17. How is the diameter of a circle found if the circumference is given?

A. Divide the circumference by π (3.1416) to get the diameter.

18. What is the radius of a circle?

A. The radius is a straight line from the center to any point on the circumference, and is equal to one-half the diameter.

19. What is meant by area?

A. Area is the surface included within any given line or lines.

20. Solve for the circumference and the area of a circle 2.4 inches in diameter.

A. Circumference = π x diameter or 3.1416 x 2.4 = 7.53984 inches. Area = π x radius squared or 3.1416 x 1.2^2 or 3.1416 x 1.44 = 4.5239 square inches.

21. What is an arc? What is a chord?

A. An arc is a portion of a curved line. A chord is a straight line joining two points of a curve, especially two points on the circumference of a circle.

22. What is meant by a segment of a circle?

A. A segment is a part of a circle bounded by an arc and its chord.

23. What is a sector?

A. A sector is a figure bounded by the arc of a plane curve and the radii drawn from the extremities of the arc.

24. How many degrees are there in a circle? minutes in a degree? seconds in a minute?

A. There are 360 degrees in a circle; 60 minutes in one degree; 60 seconds in one minute; and 3600 seconds in one degree.

25. Give the symbols for degree, minute, and second.

A. The degree is indicated by $^\circ$, the minute by ', and the second by ". For example, an angle of 23 degrees, 17 minutes, and 13 seconds is written $23^\circ 17' 13"$.

26. Reduce $2^\circ 52' 30"$ to degrees.

A. Change the minutes and seconds in the given angle to seconds, and divide by 3600". 52' = 52 x 60" = 3120". 3120"+ 30" = 3150". 3150" ÷ 3600 = $.875^\circ$. Adding the original number of degrees, $2^\circ + .875 = 2.875^\circ$. If the angle is given in degrees and minutes, divide the number of minutes by 60' to get the number of degrees.

27. Reduce 11.5625° to degrees, minutes, and seconds.

A. Multiply .5625 by 60' to obtain the number of minutes: .5625 x 60 = 33.75'. Multiply the decimal part of the minute by 60" to obtain the number of seconds: .75 x 60 = 45". Therefore, $11.5625^\circ = 11^\circ 33' 45"$.

28. What is meant by perimeter?

A. The perimeter is the distance around a body or figure. It is equal to the sum of the sides.

29. Use "perimeter" and "circumference" in sentences.

A. The perimeter of the rectangle is 24 inches. The circumference of the circle is 6.2832 inches.

30. What is a quadrilateral?

A. A quadrilateral is a figure bounded by four straight lines, as in the sketches in Fig. 341.

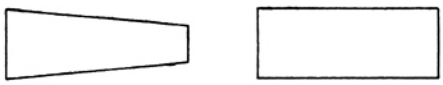


Fig. 341

31. What is a parallelogram?

A. A parallelogram is a quadrilateral with both pairs of opposite sides equal, as in the sketches in Fig. 342.

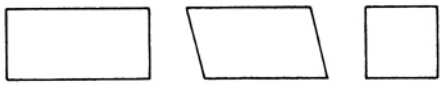


Fig. 342

32. What is a right angle? How many degrees does it contain?

A. Right angles are the angles formed when two perpendicular lines meet or intersect. Right angles contain 90 degrees.

33. What is a rectangle?

A. A rectangle is a parallelogram having four right angles, as in Fig. 343. The area of a rectangle is equal to the length multiplied by the width.

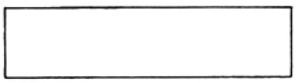


Fig. 343

34. What is a square? How is its area found?

A. A square is a rectangle having four equal sides and four right angles, as in Fig. 344. Its area is equal to the square of one side.

Fig. 344

35. What is a triangle? How is its area found?

A. A triangle is a plane figure bounded by three straight lines and having three angles. Its area is equal to one-half the base multiplied by the altitude.

36. What is a right triangle? What is the hypotenuse of a right triangle?

A. A right triangle is a triangle in which one of the angles is 90 degrees. The hypotenuse is the longest side of a right triangle.

37. Make a sketch of a right triangle, showing the different parts.

A. See Fig. 345.

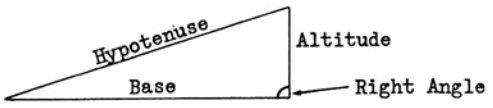


Fig. 345

38. How can the length of the hypotenuse be found when the lengths of the other two sides are known?

A. The square of the hypotenuse is equal to the sum of the squares of the other two sides. For example, find the length of the hypotenuse of a right triangle if the base is four inches and the altitude is three inches. $3^2 + 4^2 = 9 + 16 = 25$. $\sqrt{25} = 5$".

39. How many threads per inch are there on a one inch micrometer screw?

A. All micrometer screws, including the one inch, have 40 threads per inch.

40. What is the difference between a prick punch and center punch?

A. The included angle of the point of a prick punch is from 30° to 55° while the included angle of a center punch is approximately 90°.

41. Give the uses of the prick punch and center punch.

A. The prick punch is used to mark scribed or layout lines. The center punch is used to mark the center of holes for drilling.

42. What is the difference between a heavy duty and a light duty screw driver?

A. The shank of the heavy duty screw driver is made square so that a wrench may be applied to it to assist in turning a screw. The shank of the light duty screw driver is usually round, because enough pressure to turn a screw can be applied at the handle.

43. Draw a sketch showing the correct way to grind a screw driver.

Fig. 346

A small land should be left on the tip of the blade (see Fig. 346). Never grind the tip of the blade as you would grind a chisel, because this will round the slot in the screw head. In grinding the point, be careful not to overheat and anneal it.

44. What angle should be ground on the point of a cold chisel?

A. The point on a chisel for cutting soft metal should be ground to an included angle of 55 to 70 degrees. If the chisel is to be used for cutting hard steel and chilled cast iron, the point should be ground to an included angle of 80 to 90 degrees.

45. What is meant by the word "lubricate"?

A. Lubricate means to apply a thin film of a friction-resisting substance (oil or grease) between moving parts.

46. What is the correct cutting angle to grind on a drill for ordinary work?

A. A cutting angle of 59 degrees (Fig. 347) should be ground on a drill used for ordinary purposes.

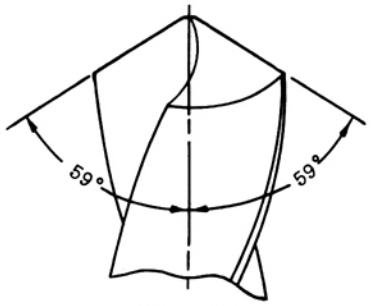


Fig. 347

47. What is the correct clearance angle to grind on a drill?

A. Grind a clearance angle of 9 to 12 degrees for hard steel and 12 to 15 degrees (Fig. 348) for soft steel. An angle of 12 degrees is generally used.

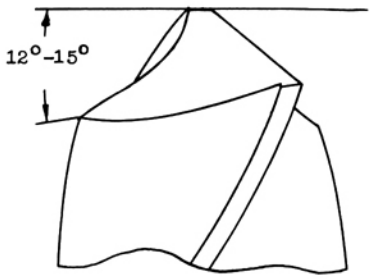


Fig. 348

48. What is the name of the taper shank used on drills?

A. Morse taper shanks are used on drills.

49. What two shapes of files are most commonly used in the machine shop?

A. The flat and the half-round files are most commonly used.

50. What kind of file should be used for filing soft metals?

A. A Vixen file should be used for filing soft metal.

51. When should a file be used without a handle?

A. Never use a file without a handle.

52. How is the single depth (Fig. 349) of an American National thread found?

A. Divide .6495 by the number of threads per inch.

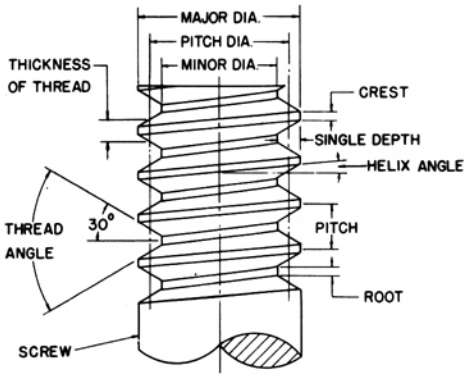


Fig. 349

53. How is the pitch diameter of an American National thread found?

A. Subtract the single depth from the major diameter (O.D.)

$$\text{P.D.} = \text{major diameter} - \frac{.6495}{\text{No. of threads per in.}}.$$

54. What is the purpose of a thread micrometer?

A. The thread micrometer is used to measure the pitch diameter of threads.

55. What included angle is ground on tools for cutting American National threads?

A. A 60 degree angle should be ground on tools for cutting American National threads.

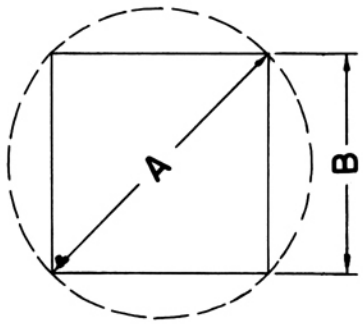
56. How is the tap drill size for a given thread determined?

A. Subtract twice the single depth, which is the double depth, from the major diameter and add one sixth of the double depth for clearance; or from the major diameter subtract

$$\frac{1.0825}{\text{No. of threads per inch}}$$

57. What is the smallest diameter from which a 3/4" square can be cut? 3/4" hexagon?

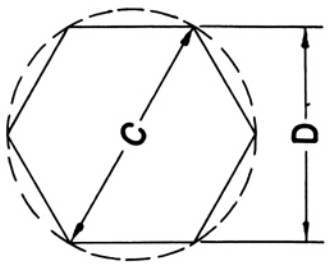
A. See Figs. 350 and 351.



$A = B \times 1.4142$

$\frac{3}{4} \times 1.4142 = 1.0606$

Fig. 350



$C = D \times 1.1547$

$\frac{3}{4} \times 1.1547 = .86602$

Fig. 351

58. Illustrate how dimensions are stated in ordering stock.

A. In writing requisitions for stock it is very important that dimensions be stated correctly and in the proper order.

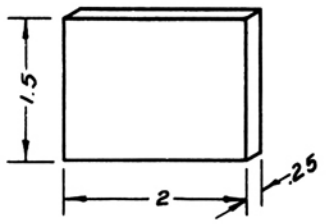


Fig. 352

In ordering rectangular stock the thickness is written first, then the width and the length. A piece of cast iron of the size shown in Fig. 352 would be stated as $\frac{1}{4}$" x $1\frac{1}{2}$" x 2".

The size of the piece of cold rolled steel shown in Fig. 353 would be stated as $\frac{1}{4}$" x $1\frac{1}{4}$" x 4".

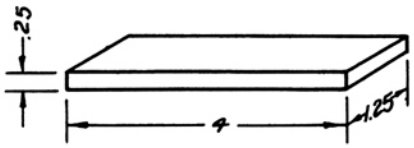


Fig. 353

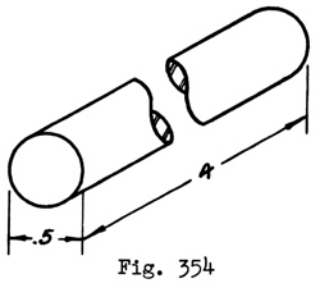


Fig. 354

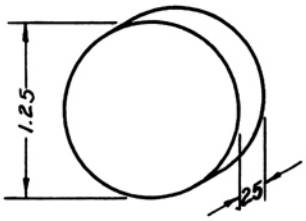


Fig. 355

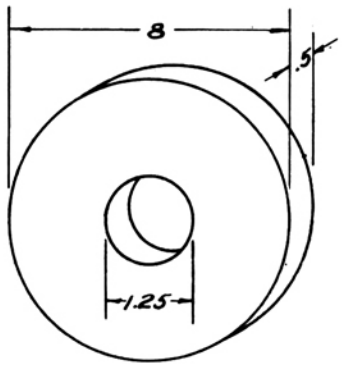


Fig. 356

In round stock the diameter is stated first, followed by the length or thickness. The size of a piece of cold rolled steel $\frac{1}{2}$" in diameter and 4" long, as shown in Fig. 354, would be written as $\frac{1}{2}$" x 4" C.R.S.

The dimensions of a piece of machine steel of the size shown in Fig. 355 would be written as $1\frac{1}{4}$" x $\frac{1}{4}$" M.S.

If round stock has a hole through it, as in milling machine cutters, grinding wheels, and cored stock, the diameter is stated first, the thickness, and then the size of hole. For example, a 46 H grinding wheel, 8" diameter and $\frac{1}{2}$" thick, with a $1\frac{1}{4}$" hole (Fig. 356), should be stated as 8" x $\frac{1}{2}$" x $1\frac{1}{4}$" 46 H.

59. What is bronze? brass?

A. Bronze is an alloy of copper and tin. Brass is an alloy of copper and zinc.

60. What is steel?

A. Steel is an alloy of iron and carbon. Among the other elements found in steel are silicon, phosphorus, sulphur, manganese, and chromium.

61. How are the different types of steel designated?

A. In Ford Motor Co. some types of steel are designated by letter. For example, "A" steel is a low carbon alloy steel; "AAA" is a medium carbon alloy steel; "RR" is a high carbon tool steel; etc. Other types are designated by name such as: chrome non-shrink, Ford hot work, insert die steel, etc. Bar stock is identified by the colors painted on the end of the bar. Many other firms use the S.A.E. numbering system for identifying steel.

62. Give the percentage of carbon in each of the following steels: cold rolled, machine, "A", "AAA", "EE", "S", and tool steel.

A. The following list gives the percentage of carbon in each of these steels.

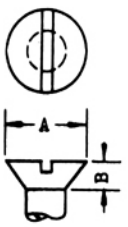
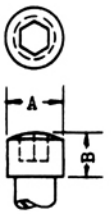
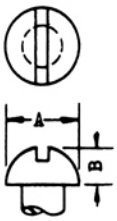
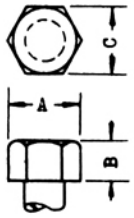
Cold rolled	.05% to .15% carbon
Machine	.08% to .20% "
"A"	.20% to .24% "
"AAA"	.30% to .35% "
"EE"	.35% to .40% "
"S"	.60% to .70% "
"RR" (Tool)	.95% to 1.05% "

63. What kind of steel is usually used to make tool bits, milling cutters, reamers, drills, broaches, etc.?

A. High speed steel is usually used for making these tools.

64. Give a table of common types of screws and tell how their lengths are measured.

A. The table below gives information on standard screws of the National Coarse and National Fine Series. The length of flat head screws is measured over the entire length. The length of socket head, button head, and hexagon head screws is measured under the head.

SIZE OF SCREW	MAJOR DIA OF THD	THDS PER INCH N C	MINOR DIA OF THD N C	THDS PER INCH N F	MINOR DIA OF THD N F	FLAT HEAD 		SOCKET HEAD 		BUTTON HEAD 		HEXAGON HEAD 		
						A	B	A	B	A	B	A	B	C
2	.086	56	.064	64	.067	.164	.045	.132	.056	.154	.060	.205	.066	.187
3	.099	48	.073	56	.077	.190	.052	.153	.063	.178	.069	.205	.066	.187
4	.112	40	.081	48	.086	.216	.060	.175	.072	.202	.078	.275	.098	.250
5	.125	40	.094	44	.097	.242	.067	.196	.081	.226	.087	.344	.114	.312
6	.138	32	.099	40	.107	.268	.075	.217	.083	.250	.096	.344	.114	.312
8	.164	32	.125	36	.130	.320	.090	.260	.106	.298	.114	.378	.130	.344
10	.190	24	.139	32	.152	.372	.105	.303	.133	.346	.133	.413	.130	.375
12	.216	24	.165	28	.172	.424	.120	.345	.141	.394	.151	.482	.193	.438
1/4	.250	20	.185	28	.206	.468	.156	.375	.250	.438	.218	.482	.242	.438
5/16	.312	18	.240	24	.261	.625	.218	.438	.312	.562	.281	.557	.250	.500
3/8	.375	16	.294	24	.324	.750	.265	.562	.375	.625	.312	.628	.289	.562
7/16	.438	14	.345	20	.375	.812	.265	.625	.438	.750	.375	.698	.337	.625
1/2	.500	13	.400	20	.438	.875	.265	.750	.500	.812	.404	.840	.385	.750
9/16	.562	12	.460	18	.494	1.00	.293	.812	.562	.938	.468	.910	.433	.812
5/8	.625	11	.507	18	.557	1.12	.360	.875	.625	1.00	.500	.980	.481	.875
3/4	.750	10	.620	16	.673	1.37	.438	1.00	.750	1.25	.625	1.12	.576	1.00
7/8	.875	9	.730	14	.787			1.12	.875			1.26	.672	1.12
1	1.000	8	.837	14	.912			1.25	1.00			1.47	.768	1.31
1-1/8	1.125	7	.939	12	1.02							1.68	.863	1.50
1-1/4	1.250	7	1.06	12	1.15							1.89	.960	1.68

Chapter 12

DRILLS AND DRILLING OPERATIONS

The twist drill shown in Fig. 357 is a very efficient tool. It is generally formed by forging and twisting grooves in a flat strip of steel or by milling a cylindrical piece of steel. High speed steel is commonly used. This steel costs more but tools made of it withstand heat much better than those made of ordinary tool steel.

The twist drill may be divided into three principal parts: body, shank, and point. The "flutes" are the spiral grooves that are formed on the side of a drill. Drills are made with two, three, or four flutes. Those having three or four flutes are used for following smaller drills or for enlarging cored holes, and are not adapted for drilling into solid stock. In fact, a drill with three or four flutes might properly be classified as an end-cutting reamer.

1. Give four advantages of spiral flutes.
   A. (1) They give the correct rake to the lip of a drill.
      (2) They cause a chip to curl tightly within itself so that it occupys the minimum amount of space.
      (3) They form channels through which chips escape from the hole.
      (4) They allow the lubricant to flow easily down to the cutting edge.

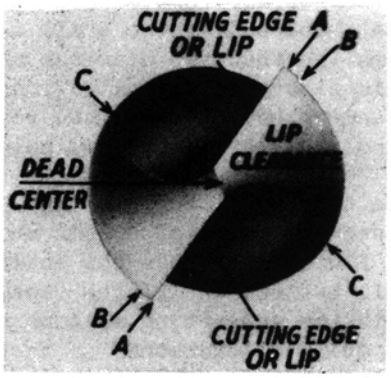
2. What is the "dead center"?
   A. The dead center is the sharp edge at the extreme tip end of the drill (see Fig. 357). It is formed by the intersection of the cone-shaped surfaces of the point and should always be in the exact center of the axis of the drill.

3. Describe the "point."
   A. The point of the drill should not be confused with the dead center. The point is the entire cone-shaped surface at the cutting end of the drill (Fig. 357).

4. Describe the "heel."
   A. The heel is the portion of the point back of the lips or cutting edges.

5. What is the "lip clearance"?
   A. The lip clearance is the surface of the point that is ground away or relieved just back of the lips (Fig. 357).

6. Describe the "margin."
   A. The narrow strip between A and B in



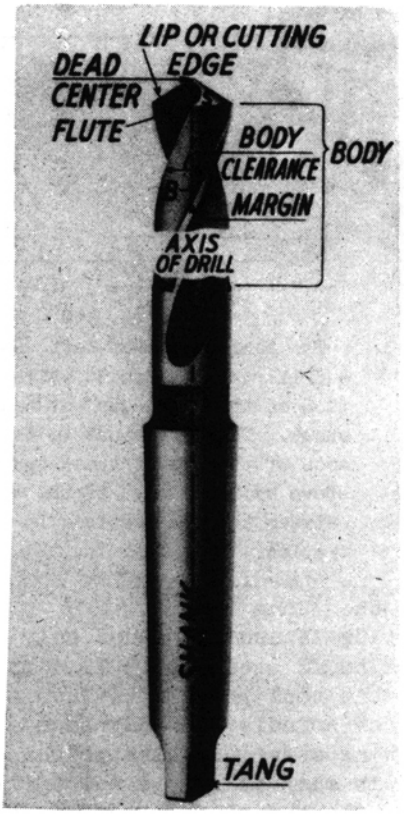


Fig. 357. Twist Drill and Point

Fig. 357 is called the margin. It is practically the full diameter of the drill and extends the entire length of the flute. Its surface is part of a cylinder which is interrupted by the flutes and by what is known as body clearance.

7. What is "body clearance"?

A. The portion from B to C in Fig. 357 is of less diameter than the margin. This lessened diameter, called body clearance, reduces the friction between the drill and the walls of the hole, while the margin between A and B (which is the full diameter) insures the hole being of accurate size. This diameter at the shank end of the drill is .0005" to .002" smaller than the diameter at the point. This clearance is called longitudinal clearance and allows the drill to revolve without binding in drilling deep holes.

8. Describe the "web" of the drill.

A. The web is the metal column which separates the flutes. It runs the entire length of the drill between the flutes (see Fig. 358), and is the supporting section of the drill, the drill's "backbone" in fact. It gradually increases in thickness toward the shank (see Fig. 359). This thickening of the web gives additional rigidity to the drill.

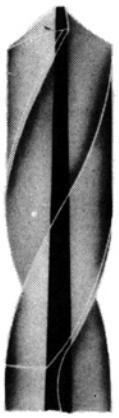


Fig. 358

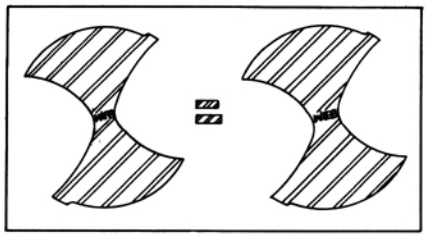


Fig. 359

The section on the left was cut from a drill near the point while the section on the right was cut near the shank. The difference in the thickness of the web at these two points is shown by the length of the white lines between the two sections in the illustration.

9. What is the "tang"?

A. The tang is usually found only on tapered shank tools (see Fig. 357). It is the portion of the tool which fits into a slot in the socket or spindle. It may bear a portion of the driving strain but its principal use is to make it easy to remove the drill from the socket with the aid of a drill drift (Fig. 360). In removing the drill, place the round edge of the drift against the sleeve and the flat edge against the top of the

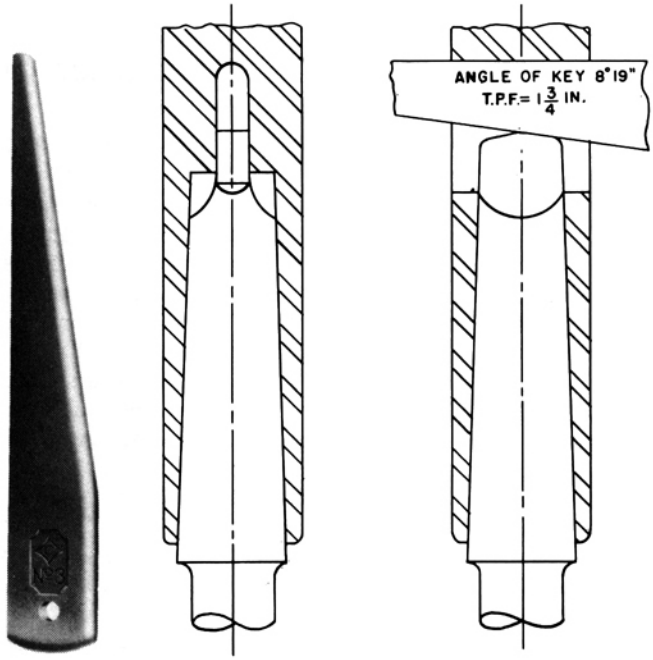


Fig. 360

drill. Never use a file or wedge as a drift. Do not use a machinist's hammer to seat the drill shank tightly in the sleeve or socket. Instead, use a piece of wood or a rawhide or lead hammer. In removing a drill from a spindle or socket, take great care that the point or tip does not hit the table of the press or the work.

10. Give some causes of broken tangs.

A. Broken tangs result from a variety of causes, such as the use of badly burred sockets, drills whose shanks are so damaged that they do not fit snugly into the sockets, or drills with badly damaged tangs.

Although the tang of a drill is designed to carry a portion of the driving strain, a tapered fit between the shank of the drill and the wall of the socket should carry a large share of this strain. A perfect fit is impossible if either the inner wall of the socket or the surface of the shank is damaged. Under such conditions the whole driving strain is thrown on the tang, and it may break through no defect of its own, but rather through faulty treatment of either the drill or the socket.

Before inserting a drill in a socket, rub off the shank to make certain it is smooth and free from grit. Also inspect the inside of the socket to be sure it is in the

same good condition.

11. What is the shank of the drill?
A. The shank is the end of the drill which fits into the socket, spindle, or chuck of the drill press. The Morse taper shank, which is the one commonly used, is shown in 357.

12. Why is the taper shank used on drills?
A. The taper shank is used so that drills may be easily forced out of a sleeve with a drill drift.

13. What makes a drill with a taper shank stick in a sleeve or spindle?
A. The taper shank is designed so that pressure at the point of the drill causes the shank to grip tighter in the sleeve, thereby keeping the drill from slipping.

14. Name the four most common shanks used on drills.
A. The four most common shanks (see Fig. 361) are the bit shank (a), straight shank (b), tapered shank (c), and ratchet shank (d).

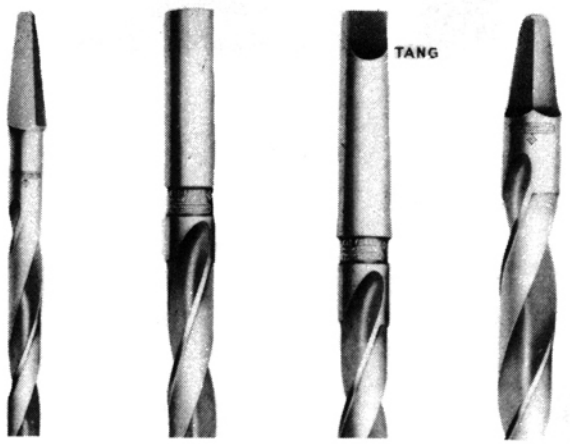


Fig. 361

15. Lips or cutting edges should be ground to what angle to the axis of a drill?
A. It has been found by experience that 59° is the best angle to grind a drill for work on steel or cast iron (see Fig. 362 in which the two lips are the same length and the same angle to the axis of the drill).

16. What happens in drilling steel or cast iron with a drill that has an angle greater or less than 59°?
A. If the angle is more than 59° the drill will not center properly, because the cone-shaped point which should hold it in

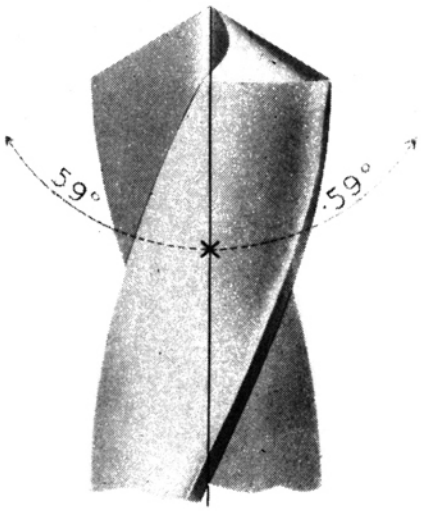


Fig. 362

position will be too flat to perform this work. If the angle is less than 59° the hole is drilled less rapidly and more power is required to drive the drill, because of the additional length of the cutting edges.

17. What happens if the point is on center but the cutting edges are ground at different angles?
A. The drill will bind on one side (see Fig. 363), only one lip or cutting edge will

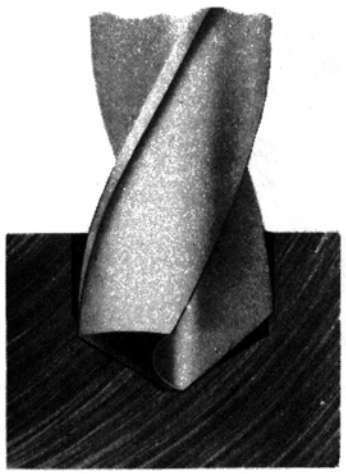


Fig. 363

do the work (resulting in rapid wear on that edge), and the hole will be larger than the drill.

18. What happens if the angles of the cutting edges are equal but the lips are of different lengths?

A. The result will be that both the point and the lip will necessarily be off center (see Fig. 364). This will cause the

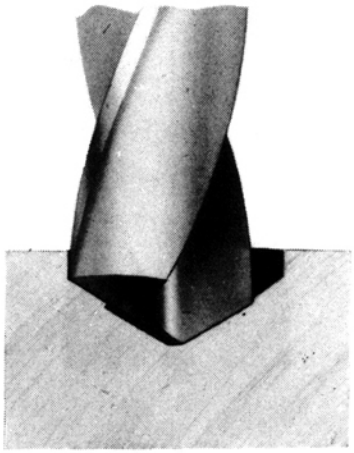


Fig. 364

hole to be larger than the drill. The effects of this condition are the same as the effects that would be obtained from a wheel with its axle placed at any point other than the exact center of the wheel. Such a condition in a drill obviously causes disastrous results. A tremendous strain is placed on the press, the spindle will weave and wobble, the drill will wear away rapidly, and the machine will eventually break down because of strains on the spindle bearings and other parts.

19. What is the approximate angle to grind the lip clearance of a drill?

A. The heel (the surface of the point back of the cutting lip) should be ground away from the cutting lip at an angle of about 12° to 15°, as shown in Fig. 365 (in all cases, this angle of 12° to 15° is the angle at the circumference of the drill).

CAUTION: Do not overheat the point

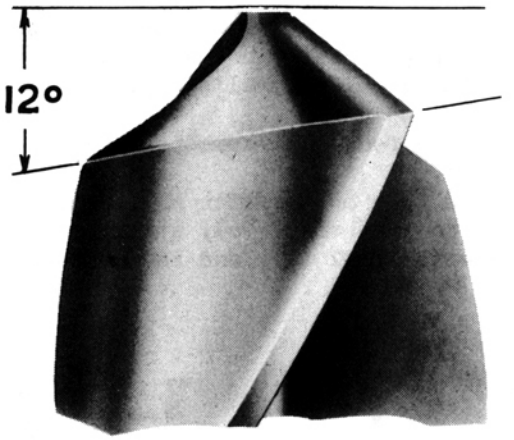


Fig. 365

Lip Clearance Angle Correctly Ground

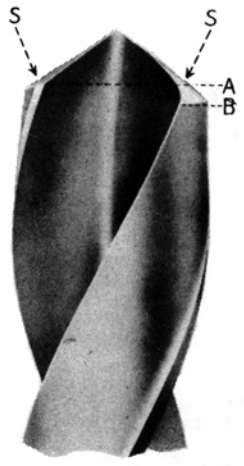


Fig. 366
Proper Lip Clearance. Note how much lower the heel line "B" is than the cutting lip line "A". This difference is the measure of the clearance.

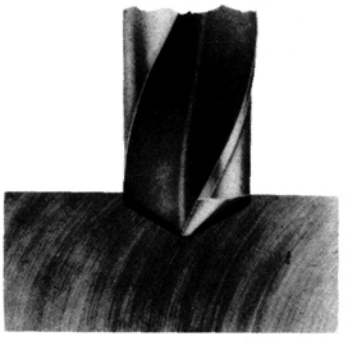


Fig. 367
The cutting lip has already removed considerable metal ahead of the heel as indicated by the black portions of the hole on each side of the drill.

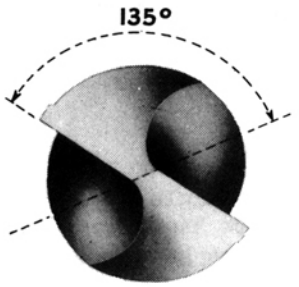


Fig. 368
Showing one way to gauge the correctness of your lip clearance angle.

of a drill in grinding it, as this will tend to anneal the cutting edge.

rolled helix.

Fig. 373 shows how chips will be removed from the job by a correctly ground drill.

Lip Clearance Angle Incorrectly Ground and Results

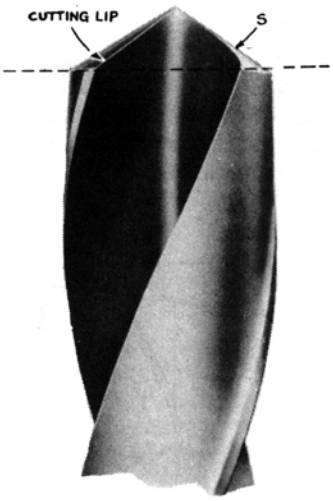


Fig. 369
Showing a drill point without any clearance. Note that both the cutting lip and the heel "S" are in the same plane.

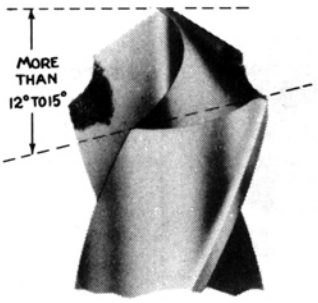


Fig. 370
Showing results of giving a drill too great lip clearance--the edges of the cutting lips have broken down because of insufficient support.

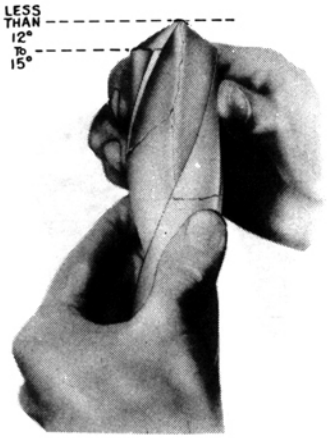


Fig. 371
Here the drill was given insufficient lip clearance. As a result there ceased to be any cutting edges whatsoever and, as the feed pressure was applied, the drill could not enter the work--as a result it "splits up the center."

20. What is the rake angle?

A. This is the angle of the flute in relation to the work (see Fig. 372). The rake angle is usually between 22° and 30°. If this angle were 90° or more, it would not give a good cutting edge. If the angle is ground too small, however, it makes the cutting edge so thin that it breaks down under the strain of the work.

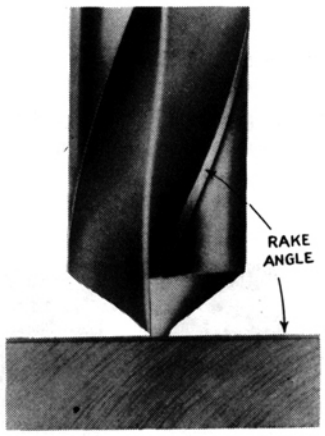


Fig. 372

The rake angle also partially governs the tightness with which the chips curl (within themselves) and hence the amount of space which the chips occupy. Other conditions being the same, a very large rake angle makes a tightly rolled chip while a rather small rake angle gives a chip a tendency to curl into a more loosely

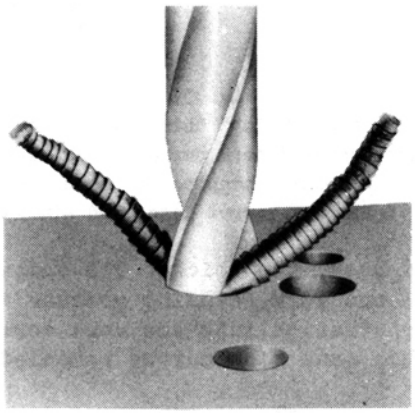


Fig. 373

21. What will happen to a drill if the speed is too great?

A. The outer corners of the drill will wear away quickly because the excessive speed will draw the temper (see Fig. 374).

22. Describe the grinding of a drill for working on brass, bronze, and cast iron.

A. Grind the lip or cutting edge so there

will be no rake, as shown in Fig. 375. When drilling brass or bronze, grind the cutting angle to 65° to 67° instead of 59°. Use a heavy feed.

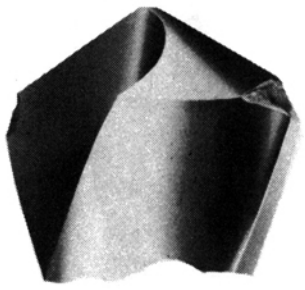


Fig. 374

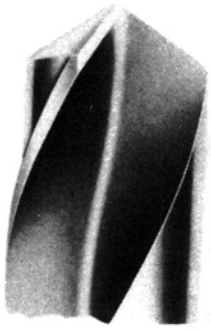


Fig. 375

23. Tell what "thinning the point" of a drill is, and explain why it is done.

A. To strengthen a drill, the thickness of the web is increased as the flute approaches the shank (Fig. 376). As the point

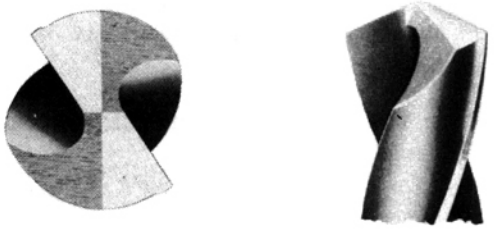


Fig. 376

Shows "notched point" thinning. This is done with a sharp cornered hard emery wheel. This type of thinning is especially adapted for hand feed such as crankshaft drilling, turret lathe and similar work.

is ground back, the thicker web comes in contact with the work, causing greater wear and making penetration into the work more difficult. This condition may be remedied by

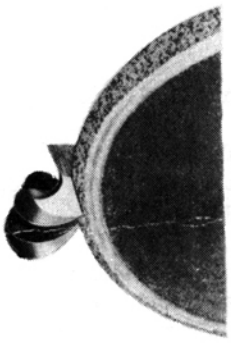


Fig. 377

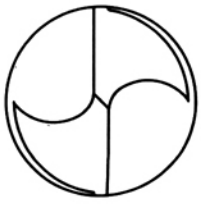


Fig. 378

thinning the point, which may be done in various ways. The use of a convex grinding wheel (Fig. 377) is the most common method. Fig. 378 shows the result of this operation.

24. Describe the grinding of a drill for working on hard material.

A. In drilling extremely hard material, using a light feed, the point should be ground to an angle of 68°, as shown in Fig. 379. The angle of lip clearance should be decreased to 9° at the periphery, as in Fig. 380. A point should be examined frequently

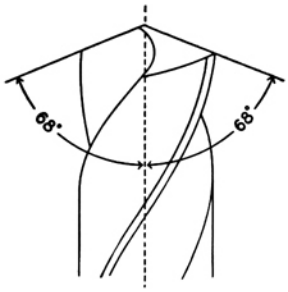


Fig. 379

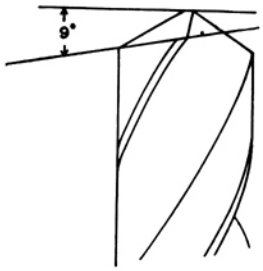


Fig. 380

as it is being ground, as a guard against drill trouble. Use turpentine, kerosene, or soda water for cooling.

NOTE. A high speed drill should be ground on a dry wheel of medium grain and soft grade. It must not be immersed in water afterward as this will cause the point of the drill to crack.

25. Name two things necessary to make a drill cut "up to size."

A. The lips must be ground exactly the same length and at the same angle to the axis.

26. What is the advantage of rotating the work in drilling a deep hole?

A. It is easier to drill a straight hole while the work is rotating, as the drill will operate like a boring bar.

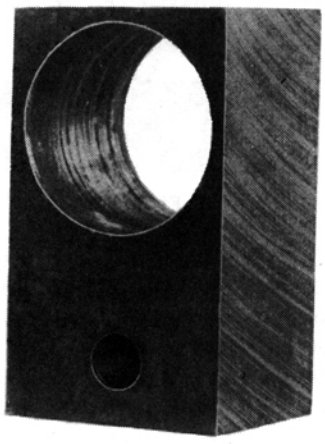


Fig. 381

27. How is a dull drill indicated?

A. The hole will be rough, as shown in Fig. 381,

and the drill will heat quickly.

28. Describe an oil hole drill.
A. The oil hole drill (Fig. 382) has oil holes through the body from the shank to the

Fig. 382

point, by which lubricant flows down to cool the point. This drill is generally used for deep hole drilling.

29. Why are flat drills used in laboratories instead of twist drills?
A. The flat drill (Fig. 383) has no rake, so that it breaks the chips into fine pieces

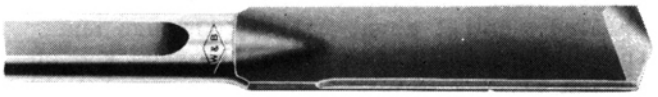


Fig. 383

suitable for laboratory work, but the twist drill causes the chips to come out of the hole in the shape of a curl or roll, as shown in Fig. 373.

30. Why is the old style flat drill, similar to the one shown in Fig. 383, better for some jobs than the modern twist drill?
A. The old style flat drill does not tend to "hog in" or "grab" soft material such as brass. Also, hard spots in steel will not throw a flat drill off center as easily as a twist drill.

31. How should the drill be started in the work?
A. After the job has been marked with a center punch (Fig. 384), the combined drill and countersink (Fig. 385) is placed in the drill chuck of a drill press and the center hole drilled.

Fig. 384

Fig. 385

In starting the drill, bring it down to the work by hand feed until it centers itself in the work. Then throw in the power feed.

32. How should a job be laid out for drilling?
A. The laying out of the holes is done from a sketch or blue print and the work is then sent to the drilling machines, ready for the necessary operations. The surface of the work to be drilled is first coated with chalk or blue vitriol. Then the height gage shown in Fig. 386 or the surface gage shown in Fig. 387 is used to locate the center of the hole to be drilled (see Fig. 390). The center punch (Fig. 384) is used to mark the location of the center and assist in starting the center drill on it. A pair of dividers (Fig. 388) is used to scribe a circle having a diameter equal to that of the hole, as shown in Fig. 391. The prick

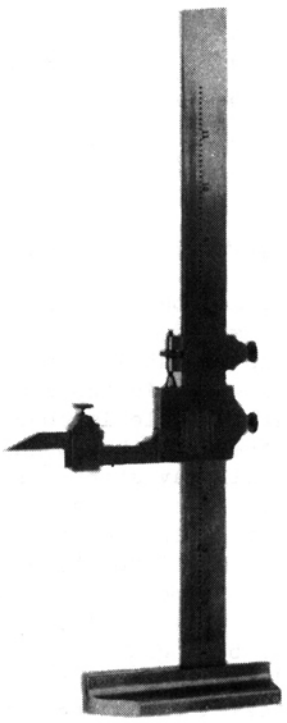
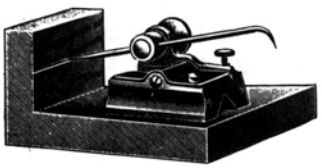


Fig. 386



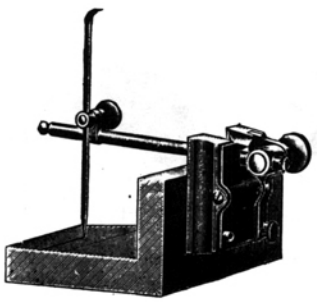


Fig. 387

Fig. 388

punch of Fig. 389 (notice that the angle of the point of the prick punch is much more acute than that of the center punch) is used to make small indentations at intervals about the circumference, as shown in Figs. 391 and 392.

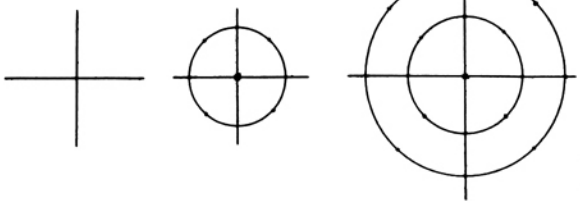


Fig. 389 Fig. 390 Fig. 391 Fig. 392

33. Explain how a drill may be drawn back on center after it has moved.

A. As a drill begins to cut into a job, it forms a conical hole which is the shape of the drill point. If the hole is concentric with the layout, the drill has been started properly. A drill may start "off center," however, due to improper center drilling, careless starting of the drill, improper grinding of the drill point, hard spots in the metal, or to some other cause (Fig. 393a). To correct this condition, use a cape chisel which has been ground with a round nose (Fig. 393d) and cut a groove on the side of the hole toward which the center is to be drawn (Fig. 393b). The amount the center has been moved may be judged by comparing the edge of the hole with the "layout line." It may be necessary to move the center several times before the edge of the hole and the "layout line" are concentric (Fig. 393c). When the drill begins to cut its full diameter, the prick punch marks on the layout should be evenly cut at the centers.

34. Why is soda water used for cooling instead of plain water?

A. The addition of soda to plain water tends to reduce the amount of heat generated, improve the finish, and overcoming rusting.

35. Why does a drill sometimes squeak?

A. The drill may be ground improperly, too hot, or running without lubrication.

36. Describe the method used to drill glass.

A. Use a piece of copper or brass tubing as a drill. The tubing should have an outside diameter of the same size as the hole desired. An abrasive is used and the tubing run at a speed of about 100 feet per minute. A coarse grinding compound or emery dust mixed with gasoline or light oil makes a very good abrasive to apply between the end of the tubing and the glass. Rest the glass on a rubber or felt pad a little larger than the size of the hole being drilled. Drill about halfway through and then invert the glass and drill from the opposite side.

37. What is the feed of a drill?

A. The distance that the drill enters the work on each revolution, measured in fractions of an inch.

38. What is meant by the speed of a drill?

A. The speed of a drill is the speed of the circumference, called peripheral speed. Speed is the distance that a drill would roll if placed on its side and rolled for one minute at its given r.p.m. (revolutions per minute). The speed of a drill is usually not expressed in revolutions per minute but in feet per minute.

39. What approximate speed in feet per minute should be used with a H.S.S. drill for alloy steel, machine steel, cast iron,

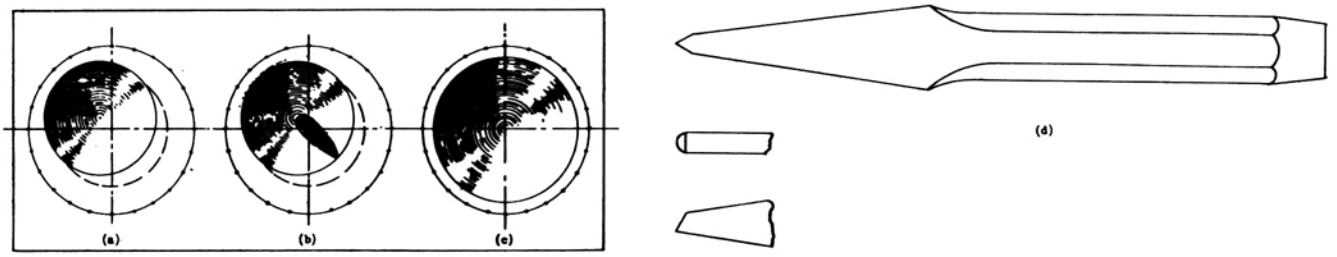


Fig. 393

and brass?

A. The following speeds are recommended by the Cleveland Twist Drill Co.

Alloy Steel	50 - 70 f.p.m.
Machine Steel	70 - 100 f.p.m.
Cast Iron	70 - 150 f.p.m.
Brass	200 - 300 f.p.m.

40. What methods are used to determine the approximate r.p.m. at which to run a $\frac{1}{4}$" drill in cast iron?

A. The correct r.p.m. may be determined mathematically as follows:

Speed for cast iron = 70 f.p.m.
70 f.p.m. = 840 inches per minute
π x diameter = circumference of drill
3.1416 x $\frac{1}{4}$ = .7854 distance per revolution
840 ÷ .7854 = 1069.51 r.p.m.

The correct r.p.m. may also be found from a table of cutting speeds. For example, if we look under 70' and opposite $\frac{1}{4}$" in the table given here we find 1070 r.p.m. Similar tables are found in various handbooks.

41. What different terms are used for sizes of wire drills and machinist's drills?

A. Wire drills are given in figure and letter sizes, as No. 46 or K. Machinist's drills are expressed in fractional sizes of an inch in diameter, as $\frac{1}{4}$" or $\frac{1}{2}$".

42. Besides the drilling of holes, what seven operations are commonly performed on the drilling machine?

A. Tapping, reaming, lapping, countersinking, counterboring, center drilling, and spot facing.

43. Describe what is meant by tapping, reaming, lapping, countersinking, counterboring, center drilling, and spot facing.

A. Taps operated by drilling machines are classified as straight, spiral, taper, pipe, and special. They are also known by several different names which indicate their use, as taper, plug, bottoming, etc.

Taps are internal threading tools. When a set of three taps (Fig. 394) is used, the taper tap makes the first cut; the plug tap (tapered only at the point) completes the thread nearly to the bottom of the hole; and the bottoming tap (which has no taper) finishes it. If the hole passes entirely through the piece and its whole length may be threaded, either the taper tap or the plug tap alone may be used.

Table of Cutting Speeds

(Fraction Size Drills)

Feet per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
Diameter Inches	Revolutions per Minute												
1/16	1833	2445	3056	3667	4278	4889	5500	6111	6722	7334	7945	8556	9167
1/8	917	1222	1528	1833	2139	2445	2750	3056	3361	3667	3973	4278	4584
3/16	611	815	1019	1222	1426	1630	1833	2037	2241	2445	2648	2852	3056
1/4	458	611	764	917	1070	1222	1375	1528	1681	1833	1986	2139	2292
5/16	367	489	611	733	856	978	1100	1222	1345	1467	1589	1711	1833
3/8	306	407	509	611	713	815	917	1019	1120	1222	1324	1426	1528
7/16	262	349	437	524	611	698	786	873	960	1048	1135	1222	1310
1/2	229	306	382	458	535	611	688	764	840	917	993	1070	1146
9/16	183	244	306	367	428	489	550	611	672	733	794	856	917
3/4	153	203	255	306	357	407	458	509	560	611	662	713	764
7/8	131	175	218	262	306	349	393	436	480	524	568	611	655
1′	115	153	191	229	267	306	344	382	420	458	497	535	573

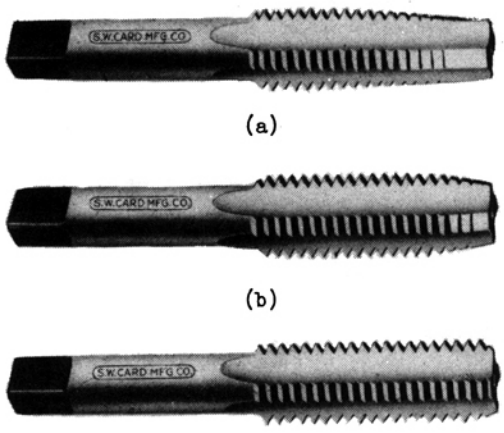


Fig. 394

Taps for internal threading: (a) taper; (b) plug; (c) bottoming.

Fig. 395 illustrates the operation of tapping on a drilling machine. A hole of the proper size must be drilled for a certain size tap, so that the thread will not be cut too deep or too shallow. A table of tap sizes and corresponding drills is given on page 83.

A tap that has been broken in a hole may be removed by a tap extractor, as shown in Fig. 396. Remove all loose particles

Fig. 395

Fig. 396

before attempting to use an extractor, and always use care and good judgment with it, as it is a fragile tool.

If a screw or bolt has broken off in a hole, it may be removed with an easy-out, as in Fig. 397. Drill a hole in the broken screw or bolt, a little smaller than its minor diameter, and then insert the easy-out and twist it as though you were tapping with a left-hand tap. The twist causes the corkscrew-like spirals of the easy-out to grip the sides of the drilled hole, so that the broken piece can be loosened and extracted.

It must be remembered that it is practically impossible to drill a hole with a high degree of accuracy. The reamer is therefore used to remove a small amount of material, leaving the surface smooth and within very close limits.

Fig. 397

Lapping may be defined as a process of finishing the surface of a piece of work by using another piece of work (called a lap) whose surface is charged with an abrasive (see Fig. 403). Only a small amount of material (.0001" to .002") is left to be removed by a lap.

Fig. 398. Straight Reamer.

Fig. 399. Taper Reamer.

Fig. 400. Expansion Reamer

Reamer Sizes			Limits of Expansion
$\frac{1}{4}$	to	$\frac{15}{32}$	.005
$\frac{1}{2}$	to	$\frac{31}{32}$	.008
1	to	$1\frac{23}{32}$	.010
$1\frac{3}{4}$	to	$2\frac{1}{2}$	.012

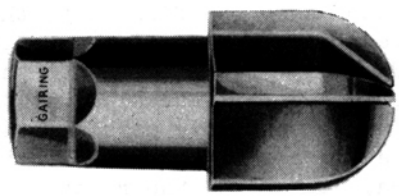


Fig. 401. Cherry Reamer.

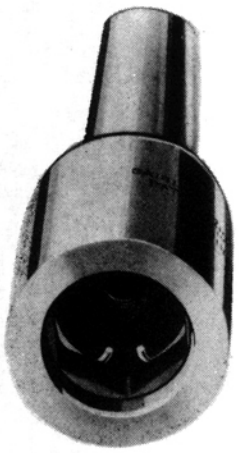


Fig. 402
End View of Holder for Cherry Reamer

Fig. 403. Copper Head Laps

Fig. 404
Countersink

The countersink in Fig. 404 is used for beveling the mouth of a hole similar to the one shown in Fig. 405.

The counterbore in Fig. 408 is used to machine a larger hole on the same center line and on top or concentric with a hole previously drilled, as shown in Fig. 406.

Center drilling is the operation of drilling and countersinking a hole with a 60° countersink, both at the same operation (see Figs. 407 and 385).

Spot facing is the machining of a circular surface around the top

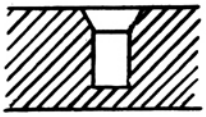


Fig. 405

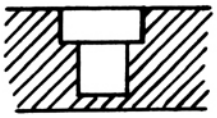


Fig. 406

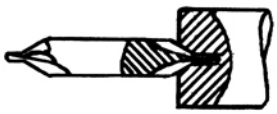


Fig. 407

of a hole to form a true bearing for a collar or washer. The counterbore shown in Fig. 408 is used for this operation.

The drilling machine is the second oldest known machine tool, having been invented shortly after the lathe, and is probably the most used of any machine. The drilling machine may be classified into three general types; Vertical Spindle, Multiple Spindle, and Radial Spindle machines.

The Vertical Spindle Drilling Machine comes in three types, Heavy Duty, plain, and Sensitive. The Heavy Duty Drilling Machine (Fig. 409) is a heavy and powerful machine for heavy drilling. It has an adjustable knee firmly gibbed to the front of the column and supported by an adjusting screw.

The Plain Vertical Spindle Drilling Machine (Fig. 410) is designed for lighter work and is more adaptable than the heavy duty machine. The spindle can be moved up and down on the column and the table can be adjusted to any desired height. These spindles are sometimes arranged in groups of three or more and called gang drills (Fig. 412). On the plain machine the table is located on an arm attached to the column and can be swung in a 180° arc at right angles to

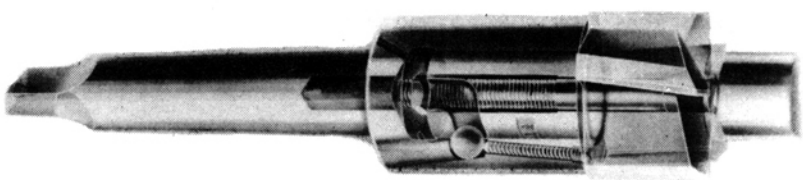


Fig. 408. Phantom View of Holder and Counterbore.

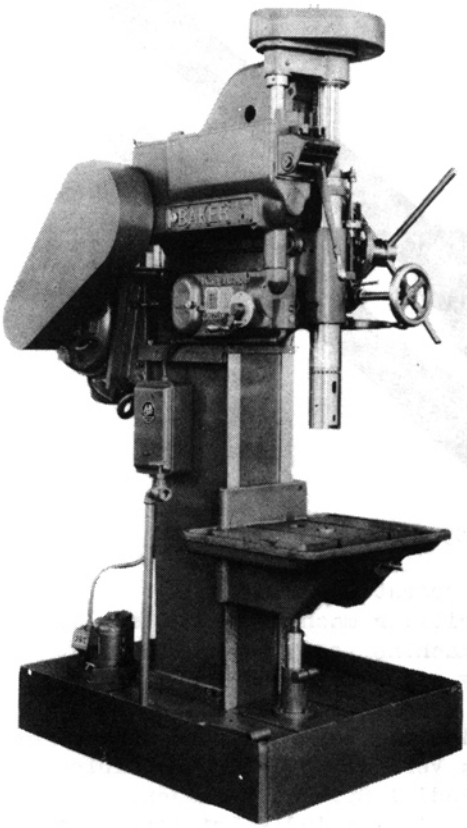


Fig. 409
Heavy Duty Drilling Machine

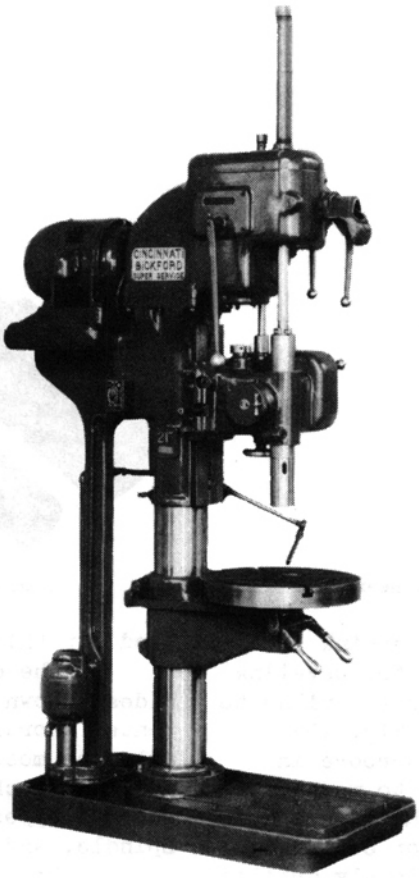


Fig. 410
Plain Drilling Machine

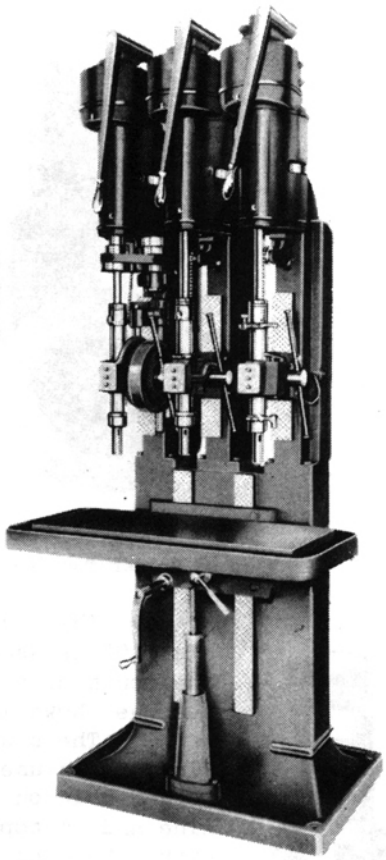


Fig. 411
Sensitive Drilling Machine

the column. The arm may be clamped in any position within the arc. The table proper is free to revolve 360° and may be clamped in any position, making it easy to locate a layout hole in line with the spindle.

The Sensitive Drilling Machine (Fig. 411) is a light drilling machine for drilling small holes, which have to be drilled at a high speed in light jobs. These machines have no feed mechanism and must be fed by hand. The table and spindle can be adjusted to the required height on the column.

The Multiple Spindle Drilling Machine (Fig. 413) may have any number of spindles arranged on the cross rail of the drilling machine, all spindles being driven from the same shaft by a worm and worm gear. The spindles are removable on the rail to accommodate different classes of work. The Multiple Spindle Drilling Machine should be distinguished from the Multiple Drill Head Machine (Fig. 414), which has any number of spindles from four to forty-eight, driven off the same spindle drive gear in the same head.

On the Radial Drilling Machine (Fig. 415), the spindle is movable and can be adjusted to the work instead of adjusting the work to fit the spindle. In cases where the work is too large and heavy to be set on the vertical spindle machine, or where a great many holes are required, the Radial Drilling Machine will be found a great convenience.

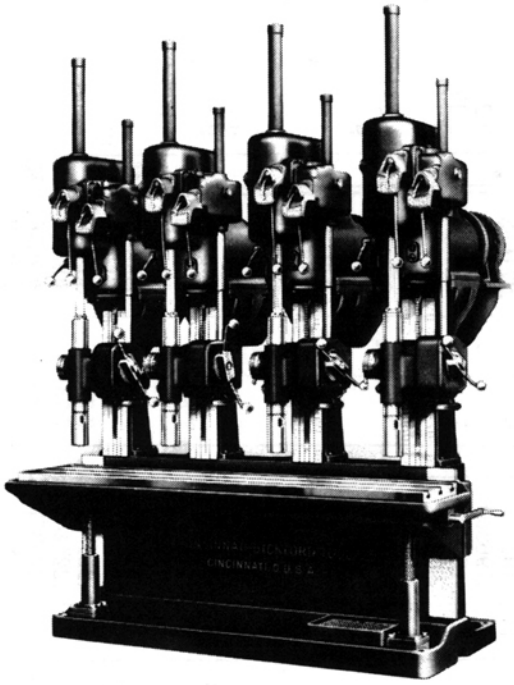


Fig. 412. Gang Drills.

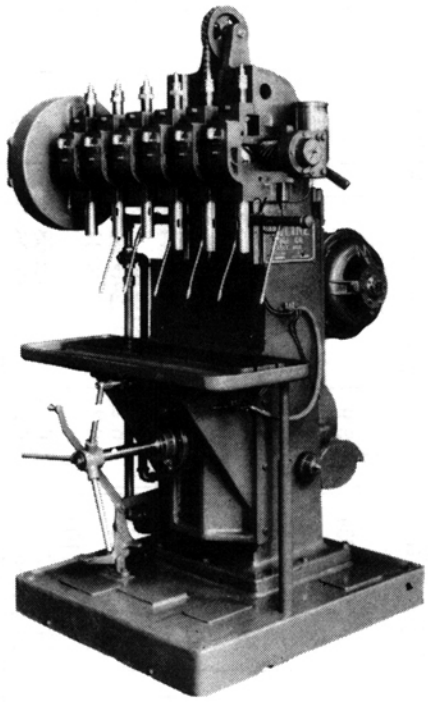


Fig. 413. Multiple Spindle Drilling Machine.

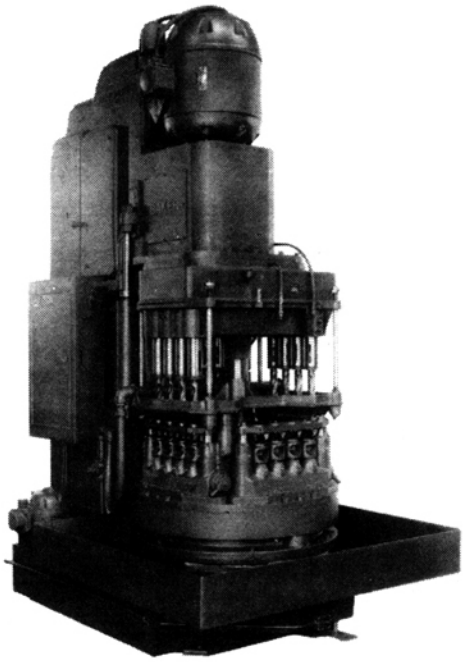


Fig. 414. Multiple Drill Head Machine.

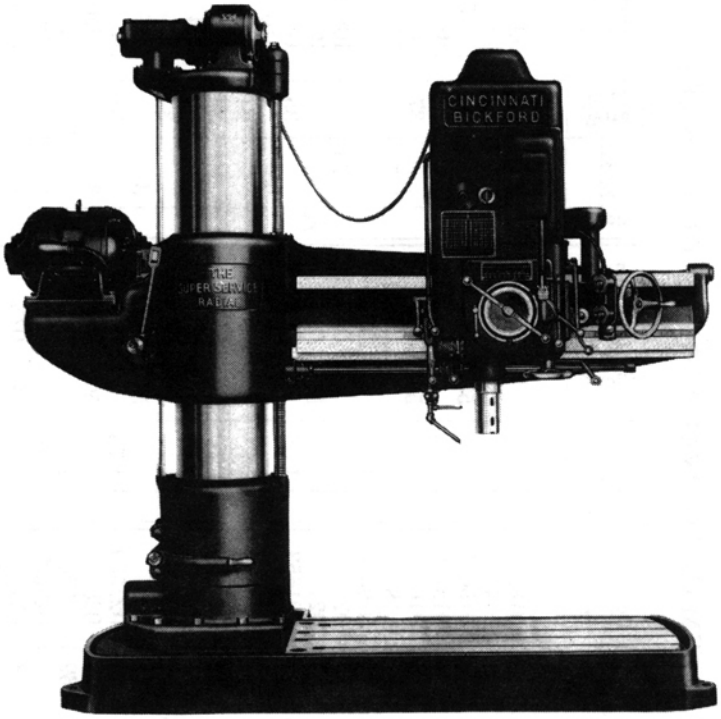


Fig. 415. Radial Drilling Machine.

DECIMAL EQUIVALENTS OF DRILL SIZES

Inch	M.M.	Wire Gauge	Decimals of an Inch
		80	.0135
		79	.0145
1/64			.015625
	.4		.015748
		78	.016
		77	.018
	.5		.019685
		76	.02
		75	.021
	.55		.021653
		74	.0225
	.6		.023622
		73	.024
		72	.025
	.65		.02559
		71	.026
	.7		.027559
		70	.028
		69	.02925
	.75		.029527
		68	.031
1/32			.03125
	.8		.031496
		67	.032
		66	.033
	.85		.033464
		65	.035
	.9		.035433
		64	.036
		63	.037
	.95		.037401
		62	.038
		61	.039
	1		.03937
		60	.04
		59	.041
	1.05		.041338
		58	.042
		57	.043
	1.1		.043307
	1.15		.045275
		56	.0465
3/64			.046875
	1.2		.047244

Inch	M.M.	Wire Gauge	Decimals of an Inch
	1.25		.049212
	1.3		.051181
		55	.052
	1.35		.053149
		54	.055
	1.4		.055118
	1.45		.057086
	1.5		.059055
		53	.0595
	1.55		.061023
1/16			.0625
	1.6		.062992
		52	.0635
	1.65		.06496
	1.7		.066929
		51	.067
	1.75		.068897
		50	.07
	1.8		.070866
	1.85		.072834
		49	.073
	1.9		.074803
		48	.076
	1.95		.076771
5/64			.078125
		47	.0785
	2		.07874
	2.05		.080708
		46	.081
		45	.082
	2.1		.082677
	2.15		.084645
		44	.086
	2.2		.086614
	2.25		.088582
		43	.089
	2.3		.090551
	2.35		.092519
		42	.0935
3/32			.09375
	2.4		.094488
		41	.096
	2.45		.096456
		40	.098

Inch	M.M.	Wire Gauge	Decimals of an Inch
	2.5		.098425
		39	.0995
		38	.1015
	2.6		.102362
		37	.104
	2.7		.106299
		36	.1065
	2.75		.108267
7/64			.109375
		35	.11
	2.8		.110236
		34	.111
		33	.113
	2.9		.114173
		32	.116
	3		.11811
		31	.12
	3.1		.122047
1/8			.125
	3.2		.125984
	3.25		.127952
		30	.1285
	3.3		.129921
	3.4		.133858
		29	.136
	3.5		.137795
		28	.1405
9/64			.140625
	3.6		.141732
		27	.144
	3.7		.145669
		26	.147
	3.75		.147637
		25	.1495
	3.8		.149606
		24	.152
	3.9		.153543
		23	.154
5/32			.15625
		22	.157
	4		.15748
		21	.159
		20	.161
	4.1		.161417

DECIMAL EQUIVALENTS OF DRILL SIZES

Inch	M.M.	Wire Gauge	Decimals of an Inch
	4.2		.165354
		19	.166
	4.25		.167322
	4.3		.169291
		18	.1695
11/64			.171875
		17	.173
	4.4		.173228
		16	.177
	4.5		.177165
		15	.18
	4.6		.181102
		14	.182
		13	.185
	4.7		.185039
	4.75		.187007
3/16			.1875
	4.8		.188976
		12	.189
	-	11	.191
	4.9		.192913
		10	.1935
		9	.196
	5		.19685
		8	.199
	5.1		.200787
		7	.201
13/64			.203125
		6	.204
	5.2		.204724
		5	.2055
	5.25		.206692
	5.3		.208661
		4	.209
	5.4		.212598
		3	.213
	5.5		.216535
7/32			.21875
	5.6		.220472
		2	.221
	5.7		.224409
	5.75		.226377
		1	.228
	5.8		.228346

Inch	M.M.	Letter Sizes	Decimals of an inch
	5.9		.232283
		A	.234
15/64			.234375
	6		.23622
		B	.238
	6.1		.240157
		C	.242
	6.2		.244094
		D	.246
	6.25		.246062
	6.3		.248031
1/4		E	.25
	6.4		.251968
	6.5		.255905
		F	.257
	6.6		.259842
		G	.261
	6.7		.263779
17/64			.265625
	6.75		.265747
		H	.266
	6.8		.267716
	6.9		.271653
		I	.272
	7		.27559
		J	.277
	7.1		.279527
		K	.281
9/32			.28125
	7.2		.283464
	7.25		.285432
	7.3		.287401
		L	.29
	7.4		.291338
		M	.295
	7.5		.295275
19/64			.296875
	7.6		.299212
		N	.302
	7.7		.303149
	7.75		.305117
	7.8		.307086
	7.9		.311023
5/16			.3125

Inch	M.M.	Letter Sizes	Decimals of an Inch
	8		.31496
		O	.316
	8.1		.318897
	8.2		.322834
		P	.323
	8.25		.324802
	8.3		.326771
21/64			.328125
	8.4		.330708
		Q	.332
	8.5		.334645
	8.6		.338582
		R	.339
	8.7		.342519
11/32			.34375
	8.75		.344487
	8.8		.346456
		S	.348
	8.9		.350393
	9		.35433
		T	.358
	9.1		.358267
23/64			.359375
	9.2		.362204
	9.25		.364172
	9.3		.366141
		U	.368
	9.4		.370078
	9.5		.374015
3/8			.375
		V	.377
	9.6		.377952
	9.7		.381889
	9.75		.383857
	9.8		.385826
		W	.386
	9.9		.389763
25/64			.390625
	10.		.3937
		X	.397
		Y	.404
13/32			.40625
		Z	.413
	10.5		.413385

Table of Cutting Speeds

(Fraction Size Drills)

Feet per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
Diameter Inches	Revolutions per Minute												
1/16	1833	2445	3056	3667	4278	4889	5500	6111	6722	7334	7945	8556	9167
1/8	917	1222	1528	1833	2139	2445	2750	3056	3361	3667	3973	4278	4584
3/16	611	815	1019	1222	1426	1630	1833	2037	2241	2445	2648	2852	3056
1/4	458	611	764	917	1070	1222	1375	1528	1681	1833	1986	2139	2292
5/16	367	489	611	733	856	978	1100	1222	1345	1467	1589	1711	1833
3/8	306	407	509	611	713	815	917	1019	1120	1222	1324	1426	1528
7/16	262	349	437	524	611	698	786	873	960	1048	1135	1222	1310
1/2	229	306	382	458	535	611	688	764	840	917	993	1070	1146
5/8	183	244	306	367	428	489	550	611	672	733	794	856	917
3/4	153	203	255	306	357	407	458	509	560	611	662	713	764
7/8	131	175	218	262	306	349	393	436	480	524	568	611	655
1″	115	153	191	229	267	306	344	382	420	458	497	535	573
1 1/8	102	136	170	204	238	272	306	340	373	407	441	475	509
1 1/4	92	122	153	183	214	244	275	306	336	367	397	428	458
1 3/8	83	111	139	167	194	222	250	278	306	333	361	389	417
1 1/2	76	102	127	153	178	204	229	255	280	306	331	357	382
1 5/8	70	94	117	141	165	188	212	235	259	282	306	329	353
1 3/4	65	87	109	131	153	175	196	218	240	262	284	306	327
1 7/8	61	81	102	122	143	163	183	204	224	244	265	285	306
2″	57	76	95	115	134	153	172	191	210	229	248	267	287
2 1/4	51	68	85	102	119	136	153	170	187	204	221	238	255
2 1/2	46	61	76	92	107	122	137	153	168	183	199	214	229
2 3/4	42	56	69	83	97	111	125	139	153	167	181	194	208
3	38	51	64	76	89	102	115	127	140	153	166	178	191

Tap Drill Sizes

Size	Threads per Inch NC	NF	NS	Outside Diameter Inches	Pitch Diameter Inches	Root Diameter Inches	Tap Drill Approx. 75% Full Thread	Decimal Equivalent of Tap Drill
0	....	80	....	.0600	.0519	.0438	3/64	.0469
1	....	....	56	.0730	.0614	.0498	54	.0550
1	64	....	....	.0730	.0629	.0527	53	.0595
1	....	72	....	.0730	.0640	.0550	53	.0595
2	56	....	....	.0860	.0744	.0628	50	.0700
2	....	64	....	.0860	.0759	.0657	49	.0730
3	48	....	....	.0990	.0855	.0719	47	.0785
3	....	56	....	.0990	.0874	.0758	45	.0820
4	....	....	32	.1120	.0917	.0714	45	.0820
4	....	....	36	.1120	.0940	.0759	44	.0860
4	40	....	....	.1120	.0958	.0795	43	.0890
4	....	48	....	.1120	.0985	.0849	42	.0935
5	....	....	36	.1250	.1070	.0889	40	.0980
5	40	....	....	.1250	.1088	.0925	38	.1015
5	....	44	....	.1250	.1102	.0955	37	.1040
6	32	....	....	.1380	.1177	.0974	35	.1065
6	....	....	36	.1380	.1200	.1019	34	.1110
6	....	40	....	.1380	.1218	.1055	33	.1130
8	....	....	30	.1640	.1423	.1207	30	.1285
8	32	....	....	.1640	.1437	.1234	29	.1360
8	....	36	....	.1640	.1460	.1279	29	.1360
8	....	....	40	.1640	.1478	.1315	28	.1405
10	24	....	....	.1900	.1629	.1359	25	.1495
10	....	....	28	.1900	.1668	.1436	23	.1540
10	....	....	30	.1900	.1684	.1467	22	.1570
10	....	32	....	.1900	.1697	.1494	21	.1590
12	24	....	....	.2160	.1889	.1619	16	.1770
12	....	28	....	.2160	.1928	.1696	14	.1820
12	....	....	32	.2160	.1957	.1754	13	.1850
1/4	20	....	....	.2500	.2175	.1850	7	.2010
1/4	....	28	....	.2500	.2268	.2036	3	.2130
5/16	18	....	....	.3125	.2764	.2403	F	.2570
5/16	....	24	....	.3125	.2854	.2584	I	.2720
3/8	16	....	....	.3750	.3344	.2938	5/16	.3125
3/8	....	24	....	.3750	.3479	.3209	O	.3320
7/16	14	....	....	.4375	.3911	.3447	U	.3680
7/16	....	20	....	.4375	.4050	.3726	25/64	.3906
1/2	13	....	....	.5000	.4500	.4001	27/64	.4219
1/2	....	20	....	.5000	.4675	.4351	29/64	.4531
9/16	12	....	....	.5625	.5084	.4542	31/64	.4844
9/16	....	18	....	.5625	.5264	.4903	33/64	.5156
5/8	11	....	....	.6250	.5660	.5069	17/32	.5312
5/8	....	18	....	.6250	.5889	.5528	37/64	.5781
3/4	10	....	....	.7500	.6850	.6201	21/32	.6562
3/4	....	16	....	.7500	.7094	.6688	11/16	.6875
7/8	9	....	....	.8750	.8028	.7307	45/64	.7656
7/8	..	14	....	.8750	.8286	.7822	13/16	.8125
7/8	....	...	18	.8750	.8389	.8028	53/64	.8281
1	8	....	....	1.0000	.9188	.8376	7/8	.8750
1	....	14	....	1.0000	.9536	.9072	15/16	.9375
1 1/8	7	....	....	1.1250	1.0322	.9394	63/64	.9844
1 1/8	....	12	....	1.1250	1.0709	1.0168	1 3/64	1.0469
1 1/4	7	....	....	1.2500	1.1572	1.0644	1 7/64	1.1094
1 1/4	....	12	....	1.2500	1.1959	1.1418	1 11/64	1.1719
1 3/8	6	....	....	1.3750	1.2667	1.1585	1 7/32	1.2187
1 3/8	....	12	....	1.3750	1.3209	1.2668	1 19/64	1.2969
1 1/2	6	....	....	1.5000	1.3917	1.2835	1 11/32	1.3437
1 1/2	....	12	....	1.5000	1.4459	1.3918	1 27/64	1.4219
1 3/4	5	....	....	1.7500	1.6201	1.4902	1 9/16	1.5625
2	4 1/2	....	....	2.0000	1.8557	1.7113	1 25/32	1.7812
2 1/4	4 1/2	....	....	2.2500	2.1057	1.9613	2 1/32	2.0313
2 1/2	4	...	....	2.5000	2.3376	2.1752	2 1/4	2.2500
2 3/4	4	....	....	2.7500	2.5876	2.4252	2 1/2	2.5000
3	4	...	. ..	3.0000	2.8376	2.6752	2 3/4	2.7500
3 1/4	4	....	.. .	3.2500	3.0876	2.9252	3	3.0000
3 1/2	4	....	....	3.5000	3.3376	3.1752	3 1/4	3.2500
3 3/4	4	....	....	3.7500	3.5876	3.4252	3 1/2	3.5000

Chapter 13

TAPERS

Most machines in the machine shop are provided with revolving spindles having tapered holes into which the tapered shanks of drills, reamers, centers, etc., are fitted and securely held in place.

The term taper may be defined as the gradual lessening or increasing of the diameter or thickness of a piece of work toward one end. The amount of taper in any given job is found by subtracting the size at the small end from the size at the large end.

Since the taper attachment on tool room lathes and grinders is usually graduated to read in either degrees or taper per foot, it is frequently necessary to calculate one of these factors. This may be done by using similar figures and ratio and proportion. It must be remembered that when using this method for solving taper problems it is necessary to know either two lengths and a taper or two tapers and a length. On blue prints the taper is usually given as an angle or as the amount of taper per foot (T.P.F.) or taper per inch (T.P.I.).

1. (a) What taper is used on drills? (b) How are taper sizes designated?

   A. (a) Drills are usually made with the Morse standard taper shank. (b) Taper sizes are designated by a numbering system, the number of the taper increasing with the size of the drill, as 1, 2, 3, 4, etc.

2. What taper is used on milling machines for the shanks of arbors, collets, end mills, etc.?

   A. The Brown and Sharpe standard taper is used for milling machine spindle shanks.

3. What standard taper does the Ford Motor Company use for the shanks of lathe centers?

   A. The Ford Motor Company uses the Jarno standard taper.

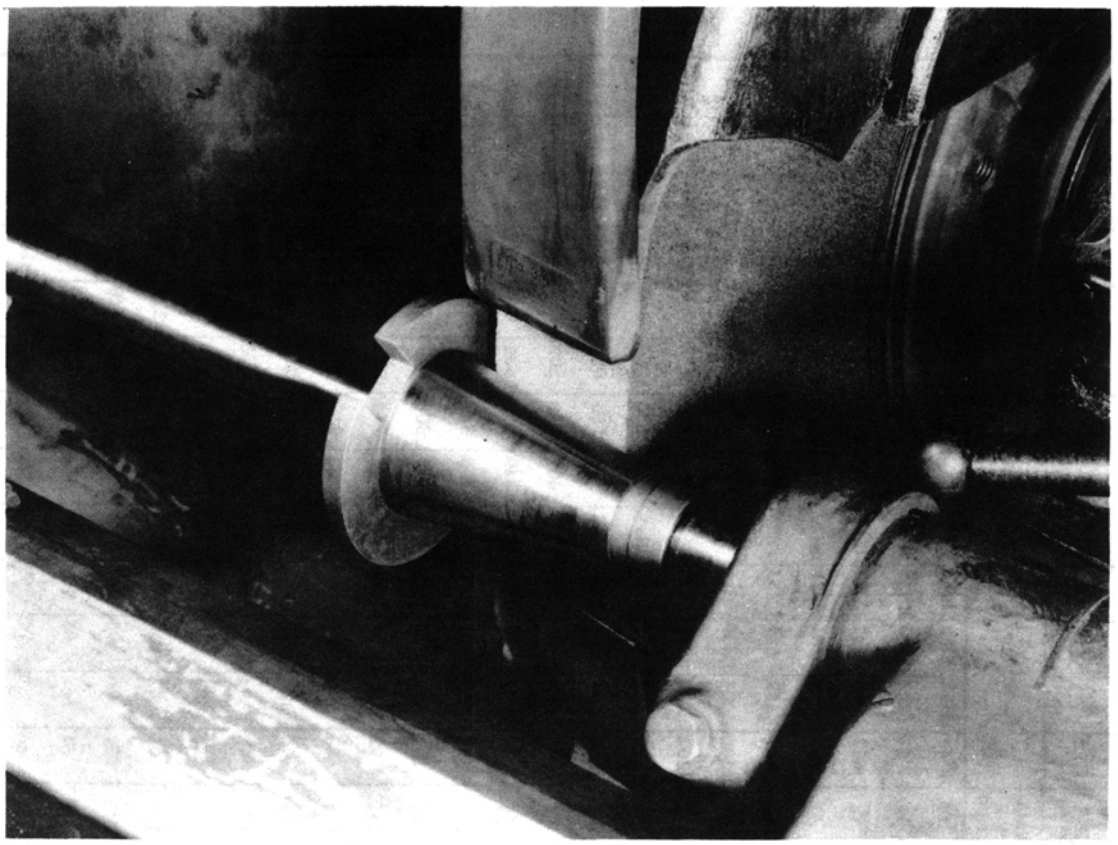


Fig. 416. Grinding the Taper on a Milling Machine Arbor

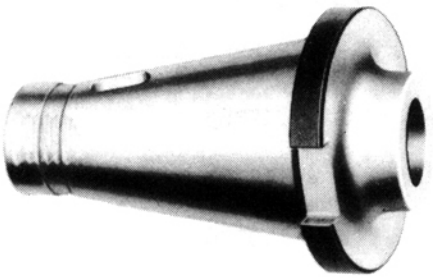


Fig. 417. Tapered Face Milling Cutter Arbor

Style 1

Style 2 Style 3

Fig. 418. Tapered Sleeve

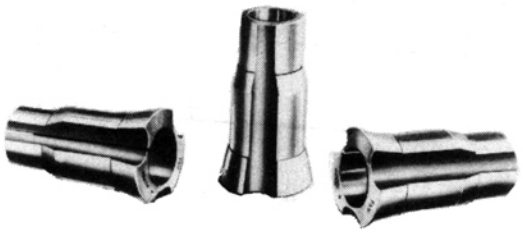


Fig. 419. Collets for Holding Tools

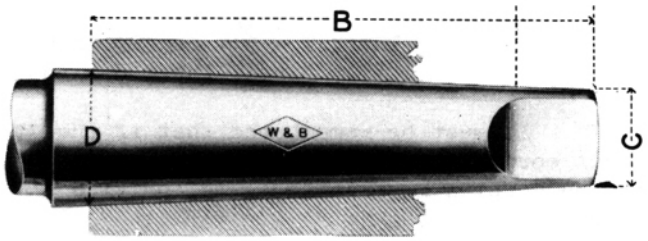


Fig. 420. Tapered Arbor for Screw Cutter

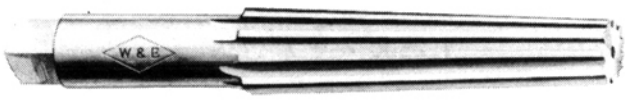


Fig. 421. Morse Standard Taper Reamer

Taper $\frac{1}{4}$ Inch per Foot

Fig. 422. Standard Taper Pin Reamer

4. What taper system is the simplest? Why?

A. The Jarno system is the simplest, because of the easy method of finding the various taper dimensions.

5. Give the Jarno taper formulas.

A.

(1) Diameter of large end = $\frac{\text{No. of Taper}}{8}$

(2) Diameter of small end = $\frac{\text{No. of Taper}}{10}$

(3) Length of taper = $\frac{\text{No. of Taper}}{2}$

(4) T.P.F. = .600 and T.P.I. = .050.

6. How may the accuracy and efficiency of tapers be preserved?

A. The accuracy and efficiency of tapers are preserved by keeping them free from dirt, chips, nicks, and burrs.

7. How should a taper ring gage be checked with a taper plug gage?

A. The taper plug gage should be covered with Prussian blue and then inserted into the ring gage, giving the plug a gentle twisting motion. Any irregularities of the bearing surface will be shown by blue marks inside the ring gage.

8. How can ratio and proportion be used in solving taper problems?

A. By using similar figures (figures which have the same characteristics but are not the same size). A ratio is an indicated division, 1/4, for example, being the ratio between 1 and 4. Where specific terms are used the ratio must be expressed between two terms of like denomination. A proportion is a statement of equality between two given ratios.

9. Find the taper per foot of the job shown in Fig. 423.

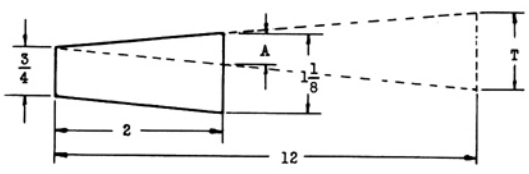


Fig. 423

The diameter of the small end subtracted from the diameter of the large end equals the amount of taper in the length of the work. Then 1-1/8 - 3/4 = 3/8, the taper in the length of the work or the distance A. Using ratio and proportion the problem may be stated as follows:

$$\frac{\text{Length}}{\text{Length}} = \frac{\text{Taper}}{\text{Taper}} \quad \text{or} \quad \frac{2}{12} = \frac{\frac{3}{8}}{T}$$

Cross multiplying, $2T = 12 \times \frac{3}{8}$

Clearing fractions, $16T = 12 \times 3$ or 36

Then, $T = \frac{36}{16}$ or $2\frac{1}{4}$ inches taper per foot

It must be remembered that if the first couplet starts with a small number the second couplet must do the same, or vice versa.

10. Find the taper per foot in Fig. 424.

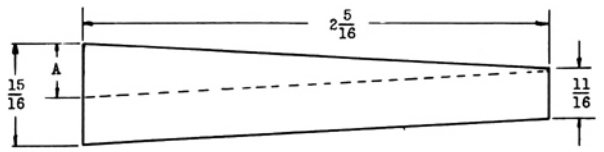


Fig. 424

$$A = \frac{15}{16} - \frac{11}{16} \quad \text{or} \quad \frac{1}{4}$$

$$\frac{2\frac{5}{16}}{12} = \frac{\frac{1}{4}}{T} \qquad 2\tfrac{5}{16}\,T = \frac{12}{4}$$

$$T = \frac{12}{4} \div 2\frac{5}{16}$$

T = 1.2972 inches taper per foot

11. Find the diameter A in Fig. 425.

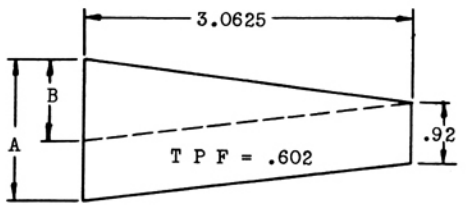


Fig. 425

$$\frac{B}{.602} = \frac{3.0625}{12}$$

$$12B = 3.0625 \times .602$$

$$B = \frac{3.0625 \times .602}{12}$$

$$B = .15363$$

$$A = .15363 + .920$$

$$A = 1.0736.$$

12. Find the distance A in Fig. 426.

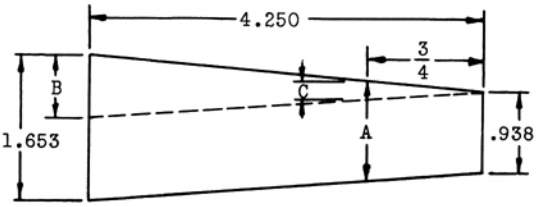


Fig. 426

$$B = 1.653 - .938 = .715$$

$$\frac{C}{.715} = \frac{\frac{3}{4}}{4.250}$$

$$4.250C = \frac{3}{4} \times .715$$

$$C = \frac{\frac{3}{4} \times .715}{4.250}$$

$$C = .12617$$

$$A = .12617 + .938$$

$$A = 1.0641.$$

13. Determine the taper per foot in Fig. 427.

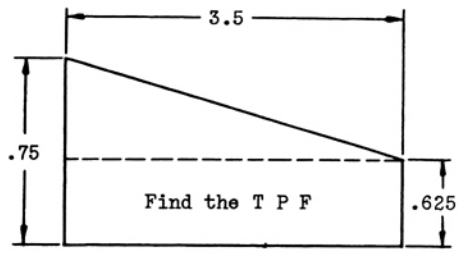


Fig. 427

$$\text{Taper} = .75 - .625 = .125$$

$$\frac{12}{3.5} = \frac{T}{.125}$$

$$3.5T = .125 \times 12$$

$$3.5T = 1.5$$

$$T = \frac{1.5}{3.5} = .42857.$$

14. Determine the diameter A in Fig. 428.

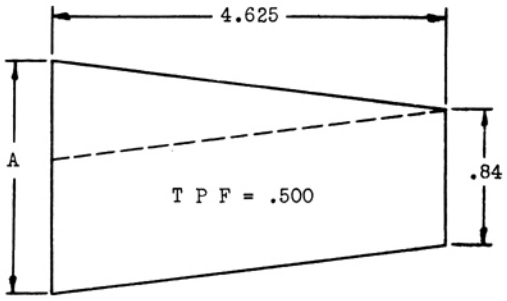


Fig. 428

$$\frac{4.625}{12} = \frac{X}{.500}$$

$$12X = 4.625 \times .500$$

$$12X = 2.3125$$

$$X = \frac{2.3125}{12} = .1927$$

$$A = .840 + .1927 = 1.0327.$$

15. Determine the diameter A in Fig. 429.

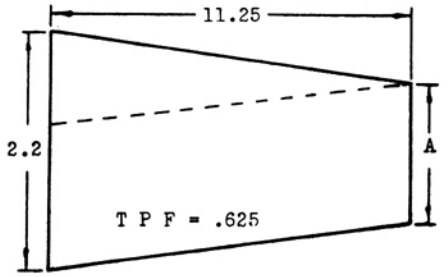


Fig. 429

Let X = taper of job

$$\frac{11.25}{12} = \frac{X}{.625}$$

$$12X = 11.25 \times .625$$

$$12X = 7.0312$$

$$X = \frac{7.0312}{12} \text{ or } .58593$$

$$A = 2.2 - .58593$$

$$A = 1.6141.$$

16. Determine the length A in Fig. 430.

Let X = taper of job

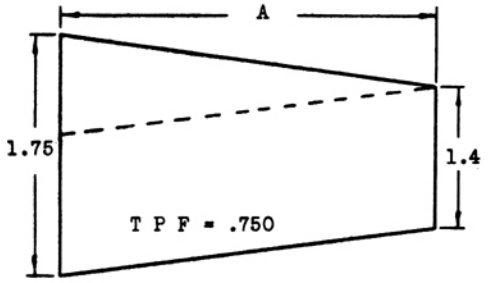


Fig. 430

$$X = 1.75 - 1.4 \text{ or } .350$$

$$\frac{.750}{.350} = \frac{12}{A}$$

$$.750A = .350 \times 12$$

$$.750A = 4.200$$

$$A = \frac{4.200}{.750}$$

$$A = 5.600.$$

17. Find the distance X in Fig. 431.

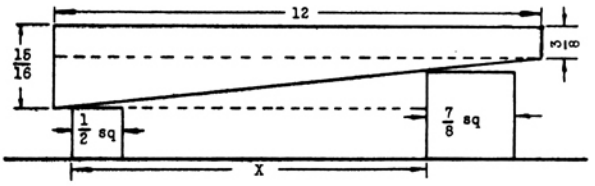


Fig. 431

$$\frac{15}{16} - \frac{3}{8} = \frac{9}{16}$$

$$\frac{7}{8} - \frac{1}{2} = \frac{3}{8}$$

$$\frac{\frac{3}{8}}{\frac{9}{16}} = \frac{X}{12}$$

$$\frac{\frac{3}{8} \times 12}{\frac{9}{16}} = 8 \qquad X = 8.$$

18. At what angle should the compound rest or taper attachment be set over if the small diameter of the work = 1.500, large diameter = 1.875, and length of taper = 6.000?

A. 1.875 - 1.500 = .375 taper in six inches

$$\frac{6}{12} = \frac{.375}{T}$$

$$T = \frac{12 \times .375}{6} \text{ or } .750 \text{ taper per foot}$$

Divide the included angle by two to find the number of degrees and minutes to set the compound rest or taper attachment. (The sixth edition of the American Machinists' Handbook, page 681, shows that a .750 taper per foot has an included angle of 3°34'44". The angle to the center line or the angle to set the taper attachment is therefore 1°47'22", which is one-half the included angle.)

19. Find the angle to set the taper attachment on a job which has a small diameter of .36875, a large diameter of .475, and a length of 2-1/8.

A. By ratio and proportion the taper per foot is found to equal .600. The taper to the center of the work will equal .600 ÷ 2 or .300. The length of 12" and the taper to the center line form a right angle triangle. Using trigonometry to find the cotangent of the angle, 12.000 ÷ .300 = 40. The cotangent of 40 equals 1°25'6", the angle to set the taper attachment.

20. Using the information on page 672 in the sixth edition of the American Machinists' Handbook, find the large diameter of a No. 5 B & S standard taper (B & S tapers have .500 taper per foot, except for No. 10 which has .5161 taper per foot).

A. Diameter at small end = .450

Length of taper = $2\frac{1}{8}$

Taper per foot = .500

Let X = taper for $2\frac{1}{8}$

$$\frac{2\frac{1}{8}}{12} = \frac{X}{\frac{1}{2}}$$

$$12X = 2\tfrac{1}{8} \times \tfrac{1}{2}$$

$$X = \frac{1.0625}{12} \text{ or } .0885$$

Diameter of plug at large end = .450 + .0885 or .5385.

21. Find the diameter at the large end of a plug with a No. 5 B & S milling machine standard taper.

A. Diameter at small end = .450

Length of taper = $1\frac{3}{4}$

Taper per foot = .500

$$\frac{1\frac{3}{4}}{12} = \frac{T}{\frac{1}{2}}$$

$$12T = 1\tfrac{3}{4} \times \tfrac{1}{2} = .875$$

$$T = \frac{.875}{12} = .0729$$

Diameter of plug at large end = .450 + .0729 = .5229.

22. If a taper plug has 3/16 of an inch taper in 4 inches, what is the taper per foot? At what angle should the taper attachment be set?

A. The equation for finding the taper per foot is

$$\frac{4}{12} = \frac{\frac{3}{16}}{T}$$

$$4T = 12 \times \frac{3}{16}$$

$$4T = 2.250$$

$$T = .5625.$$

Information on page 681 of the American Machinists' Handbook gives the included angle for .5625 taper per foot as 2°41'4". The taper attachment would be set at one-half of this angle, or at 1°20'32".

23. Give the dimensions for a No. 9 Jarno taper (see question 5).

A.

Large diameter = $\frac{9}{8} = 1.125$

Small diameter = $\frac{9}{10} = .900$

Length = $\frac{9}{2} = 4.500$

Taper per foot = .600.

24. If .015 grinding stock is left on a #7 B & S taper plug gage, how far will it come from reaching the end of the ring gage (see Fig. 432)?

A. The taper per foot for a #7 B & S taper is .500.

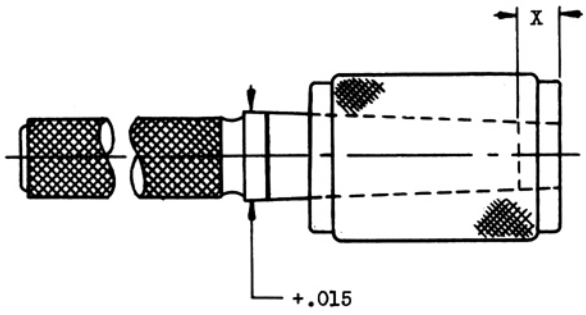


Fig. 432

$$\frac{.015}{.500} = \frac{X}{12}$$

$$.500X = .015 \times 12$$

$$.500X = .180$$

$$X = \frac{.180}{.500} = .360.$$

25. Give the amount of taper per foot for three tapers in common use.

A.

Brown Sharpe	- .500 T.P.F.	(except #10 which has .5161 T.P.F.)
Jarno	- .600 T.P.F.	
Morse	- .600 - .630 T.P.F.	(depending on the number of taper)

26. What is a taper dowel pin? What is the T.P.F. for a taper dowel pin?

A. A taper dowel pin is a pin used to hold jigs, fixtures, brackets, machine parts, etc., in alignment after they have been located properly. These pins have a taper of 1/4" per foot.

27. How may tapers be produced on a lathe?

A. Tapers may be produced on a lathe by setting over the tailstock, by setting the compound at an angle and feeding the tool by hand, by using a taper attachment, and by using a square nose tool.

28. How is a taper hole produced with a taper reamer?

A. To produce a taper hole with a reamer, first drill a hole large enough to allow the small end of the taper to enter. Then ream out by hand with a taper reamer until the proper size is reached.

29. How far must a boring tool be fed into the work to allow a .750 T.P.F. plug gage to advance .062 into a ring gage?

A. Solve for the amount of taper in .062 and divide the result by 2.

$$\frac{.750}{X} = \frac{12}{.062}$$

$$12X = .750 \times .062$$

$$X = \frac{.750 \times .062}{12} = .003875$$

$$.003875 \div 2 = .00194.$$

Chapter 14

THREADS

TERMS RELATING TO SCREW THREADS

1. Screw Thread. A ridge of uniform section in the form of a helix on the surface of a cylinder or cone.

2. External and Internal Threads. An external thread is a thread on the outside of a member. Example: A threaded plug. An internal thread is a thread on the inside of a member. Example: A threaded hole.

3. Major Diameter (formerly known as "outside diameter"). The largest diameter of the thread of the screw or nut. The term "major diameter" replaces the term "outside diameter" as applied to the thread of a screw and also the term "full diameter" as applied to the thread of a nut.

4. Minor Diameter (formerly known as "core diameter" or "root diameter"). The smallest diameter of the thread of the screw or nut. The term "minor diameter" replaces the term "core diameter" as applied to the thread of a screw and also the term "inside diameter" as applied to the thread of a nut.

5. Pitch Diameter. On a straight screw thread, the diameter of an imaginary cylinder, the surface of which would pass through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder. On a taper screw thread, the diameter, at a given distance from a reference plane perpendicular to the axis of an imaginary cone, the surface of which would pass through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cone.

6. Pitch. The distance from a point on a screw thread to a corresponding point on the next thread measured parallel to the axis.

$$\text{Pitch in inches} = \frac{1.0000}{\text{Number of threads per inch}}$$

7. Lead. The distance a screw thread advances axially in one turn. On a single-thread screw, the lead and pitch are identical; on a double-thread screw the lead is twice the pitch; on a triple-thread screw, the lead is three times the pitch, etc.

8. Angle of Thread. The angle included between the sides of the thread measured in an axial plane.

9. Helix Angle. The angle made by the helix of the thread at the pitch diameter with a plane perpendicular to the axis.

10. Crest. The top surface joining the two sides of a thread.

11. Root. The bottom surface joining the sides of two adjacent threads.

12. Side. The surface of the threads which connect the crest with the root.

13. Axis of a Screw. The longitudinal central line through the screw.

14. Base of Thread. The bottom section of the thread, the greatest section between the two adjacent roots.

15. Depth of Thread. The distance between the crest and the base of the thread measured normal to the axis.

16. Number of Threads. Number of threads in one inch of length.

17. Length of Engagement. The length of contact between two mating parts, measured axially.

18. Depth of Engagement. The depth of thread contact of two mating parts, measured radially.

19. Pitch Line. An element of the imaginary cylinder or cone specified in Definition 5.

20. Thickness of Thread. The distance between the adjacent sides of the thread measured along or parallel to the pitch line.

TERMS RELATING TO CLASSIFICATION AND TOLERANCE

Five distinct classes of screw thread fits have been established by the National Screw Thread Commission for the purpose of insuring the interchangeable manufacture of screw thread parts throughout the country. The numbers and corresponding fits are as follows: #1 - loose fit, #2 - free fit, #3 - medium fit, #4 - close fit, #5 - wrench fit.

21. Allowance. An intentional difference in the dimensions of mating parts. It is the minimum clearance or the maximum interference which is intended between mating parts. It represents the condition of the tightest permissible fit, or the largest internal member mated with the smallest external member. This is illustrated by the two following examples.

One-half inch, class 1, loose fit, American National coarse thread series:

Minimum pitch diameter of nut	- .4500
Maximum pitch diameter of screw	- .4478
Allowance (positive)	- .0022

One-half inch, class 4, close fit, American National coarse thread series:

Minimum pitch diameter of nut	- .4500
Maximum pitch diameter of screw	- .4504
Allowance (negative)	- .0004

22. Tolerance. The amount of variation permitted in the size of a part. Example:

One-half inch screw, class 1, loose fit, American National coarse thread series:

Maximum pitch diameter - -	.4478
Minimum pitch diameter - -	.4404
Tolerance - -	.0074

23. Basic Size. The theoretical or nominal standard size from which all variations are made.

24. Crest Allowance. Defined on a screw form as the space between the crest of a thread and the root of its mating thread.

25. Finish. The character of the surface of a screw thread or other product.

26. Fit. The relation between two mating parts with reference to the conditions of assembly, as wrench fit, close fit, medium fit, free fit, and loose fit. The quality of fit depends upon both the relative size and finish of the mating parts.

27. Neutral Zone. A positive allowance (see "Allowance").

28. Limits. The extreme permissible dimensions of a part. Example:

One-half inch screw, class 1, loose fit, American National coarse thread series:
Maximum pitch diameter - .4478 These are
Minimum pitch diameter - .4404 the limits.

The old U.S.S. thread has been changed to American National coarse thread series.

The old S.A.E. thread has been changed to American National fine thread series.

American National Form of Threads

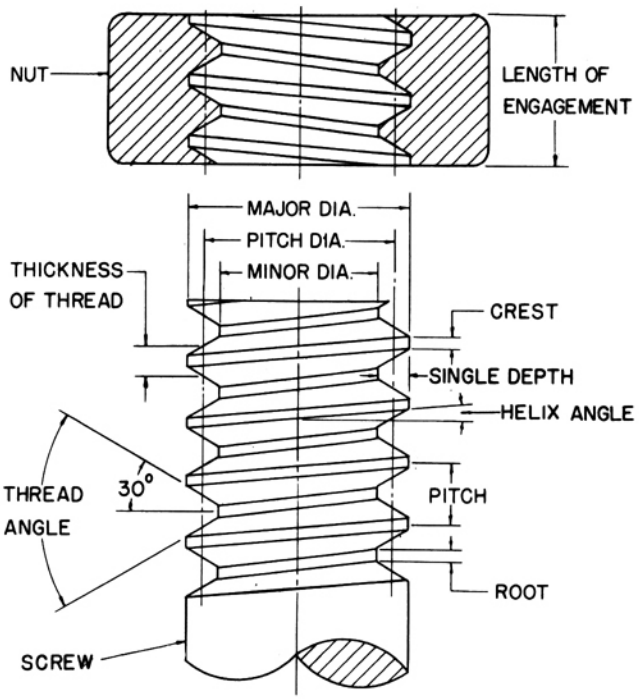


Fig. 433

N = Number of Threads per Inch

$$\text{Pitch (P)} = \frac{1.000}{N} \qquad \text{Root} = \frac{P}{8}$$

$$\text{Crest} = \frac{P}{8} \qquad \text{Single Depth} = \frac{.6495}{N}$$

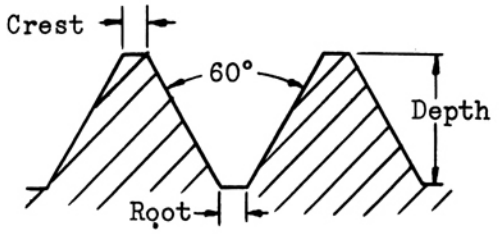
$$\text{Double Depth (D.D.)} = \frac{1.299}{N}$$

$$\text{Clearance} = \frac{D.D.}{6}$$

Pitch Diameter = Major Diameter - Single Depth
Minor Diameter = Major Diameter - Double Depth

STANDARD THREADS

AMERICAN NATIONAL STANDARD THREAD



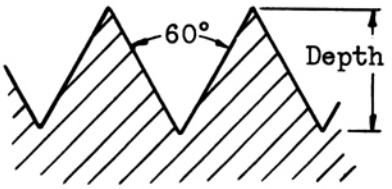
$$\text{Depth} = \frac{0.6495}{N}$$

$$\text{Crest} = \frac{\text{Pitch}}{8}$$

$$\text{Root} = \frac{\text{Pitch}}{8}$$

(a)

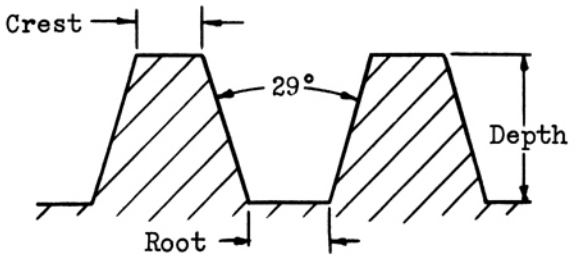
SHARP "V" THREAD



$$\text{Depth} = \frac{0.8660}{N}$$

(b)

ACME THREAD



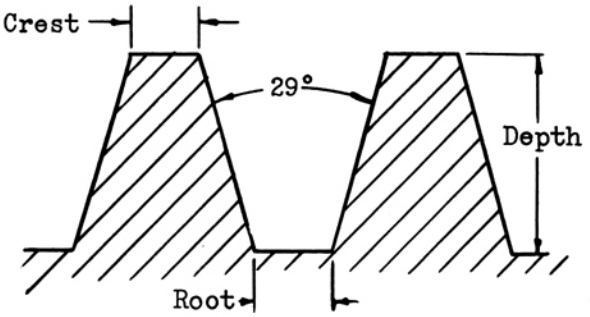
$$\text{Depth} = \frac{0.500}{N} + 0.010"$$

$$\text{Crest} = \frac{0.3707}{N}$$

$$\text{Root} = \frac{0.3707}{N} - .0052"$$

(c)

BROWN & SHARPE WORM THREAD

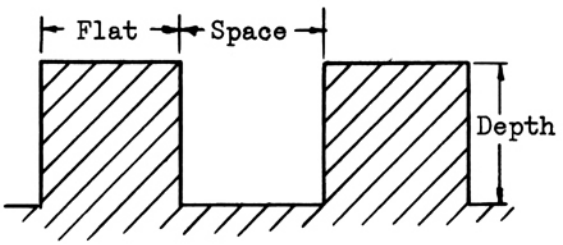


$$\text{Depth} = \frac{0.6866}{N} \qquad \text{Root} = \frac{0.310}{N}$$

$$\text{Crest} = \frac{0.335}{N}$$

(d)

SQUARE THREAD

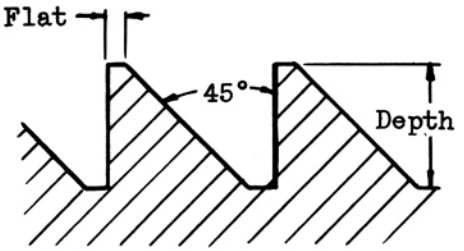


$$\text{Depth} = \frac{0.5000}{N} \qquad \text{Width of flat} = \frac{0.5000}{N}$$

$$\text{Width of space} = \frac{0.5000}{N}$$

(e)

BUTTRESS THREAD



$$\text{Depth} = \frac{0.7500}{N}$$

$$\text{Width of flat} = \frac{\text{Pitch}}{8}$$

(f)

Fig. 435

American National Form of Threads

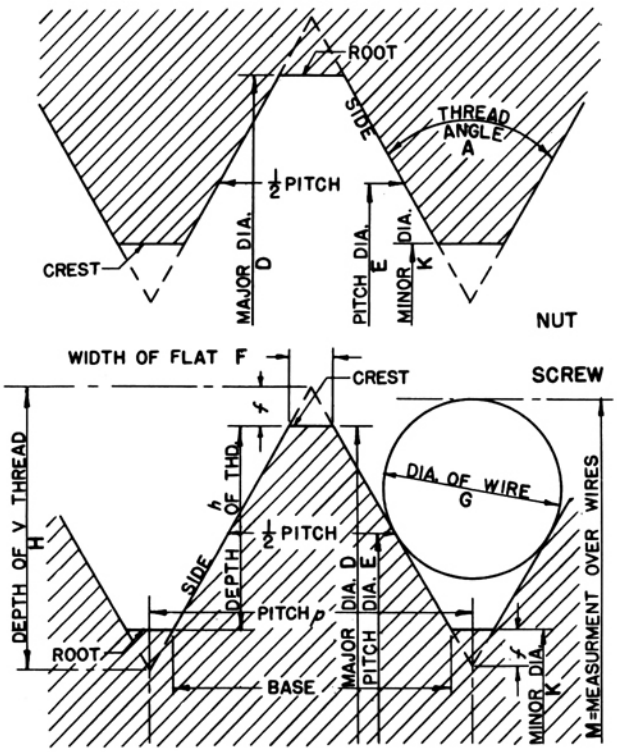


Fig. 434

Size to Bore or Drill a Hole for Internal Threading = Major Dia. $- \frac{1.0825}{N}$

Tangent of Helix Angle = $\frac{\text{Lead}}{\text{Pitch Dia. X } 3.1416}$

M (measurement over three wires) = Major Dia. $+ 3G - \frac{1.5155}{N}$

Best Wire = $\frac{.57735}{N}$ Min. Wire = $\frac{.56}{N}$

Max. Wire = $\frac{.90}{N}$

29. Describe the difference between the lead and the pitch of a screw.

A. The pitch is the distance between a point on one thread and the corresponding point on the next, measured parallel to the axis. The lead is the distance a nut will advance on a screw, parallel to its axis, in one revolution (Fig. 436). On a single thread screw the lead equals the pitch, on a double thread screw it equals twice the pitch, etc.

30. Describe the tap and its use.

A. A tap is a cylindrical bar of steel with threads formed around it and grooves or "flutes" running lengthwise in it, intersecting with the threads to form cutting edges. These cutting edges cut the threads in the nut.

Fig. 437. Taps: (a) taper; (b) plug; (c) bottoming.

31. Name three different kinds of taps.

A. Taper, plug, and bottoming taps (shown in Fig. 437).

32. State how to find the size of hole to drill for tapping a National Form thread.

A. Divide 1.0825 by the number of threads per inch and subtract this number from the major diameter.

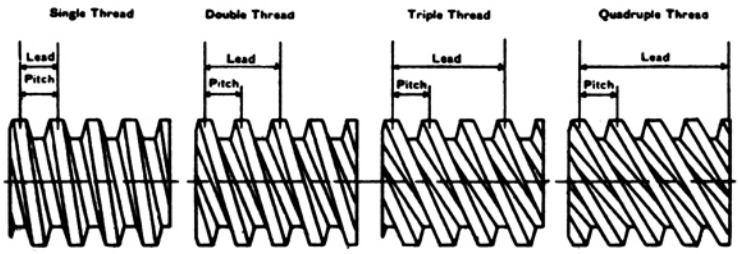


Fig. 436. Relation of Lead and Pitch of Multiple Threads

To check the squareness of the tap with the hole, place the blade of a square against the tap, permitting the beam of the square to rest on the surface of the work, as shown in Fig. 438.

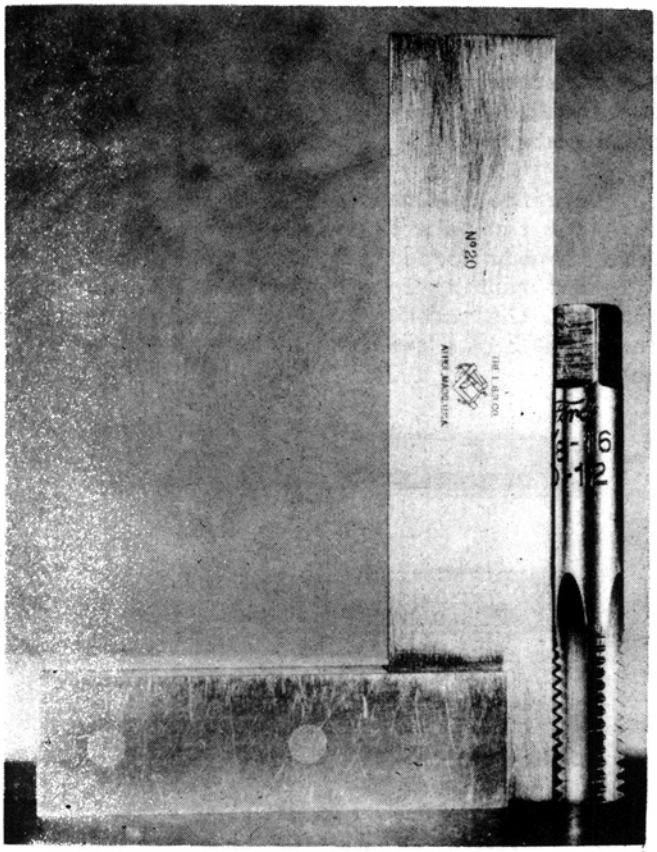


Fig. 438. Testing Tap for Squareness with Work

33. Describe a die and its use.

A. A die is usually a flat piece of steel, internally threaded, with grooves or flutes intersecting the threads to form cutting edges (Fig. 439. These cut the external threads on the screw.

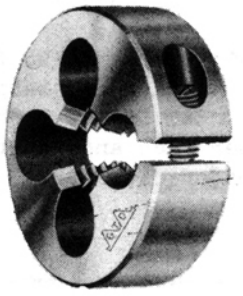


Fig. 439. Die

34. Find the size of the hole to drill to tap the following American National threads: 3/4 NC; 1/2 NF, and 3/4 - 12N. Use the following formula (N = number of threads per inch):

$$\text{Size of Hole} = \text{Major Diameter} - \frac{1.0825}{N}$$

A.

(a) .750 major dia. $- \frac{1.0825}{10}$ = .750 - .10825

= .64175

(b) .500 major dia. $- \frac{1.0825}{20}$ = .500 - .0541

= .4459

(c) .750 major dia. $- \frac{1.0825}{12}$ = .750 - .0902

= .6598.

35. Why is the Sharp "V" thread almost obsolete?

A. The sharp top of the "V" thread is very easily nicked or burred, and the taps and dies will not stand up long because of the sharp thread.

36. After the first cut is taken on a lathe, how should the correct number of threads per inch be checked?

A. Place a rule on the thread parallel with the center line and count the number of threads in one inch (or the number of threads in one-half inch and multiply by two), or check with a pitch gage.

37. What is meant by "rolled threads"?

A. Most of the external commercial threads made today are rolled, that is, the part to be threaded is rolled between grooved dies in a special machine.

38. How is the major diameter (O.D.) of the blanks for rolled threads determined?

A. The major diameter of a blank is equal to the pitch diameter of the finished thread.

39. Figure the size of the blank stock that should be used for 3/4" American National coarse rolled threads.

A. A 3/4" dia. thread has ten threads per inch. We use the following formula to find the pitch diameter: Pitch Diameter = Major Diameter - Single Depth.

$$\text{Single Depth} = \frac{0.6495}{10} = .06495$$

Major Diameter - - - - - - - - -	0.75000
Single Depth - - - - - - - - - -	.06495
Pitch Dia. of Finished Thread -	.68505

40. Find the number of threads per inch on a screw with a pitch of .750.

A.

$$\frac{1.0000}{.750} = 1\frac{1}{3} \text{ threads per inch.}$$

41. Find the lead of a screw which has 16 threads per inch.

A.

$$\text{Lead} = \frac{1.000}{16} = 0.0625$$

The lead and the pitch are the same for single threads. If this were a double thread, the lead would be 0.125.

42. Figure the minor diameter of a Brown & Sharpe triple worm thread with 3/4" lead and 3.000" major diameter.

A. The pitch equals the lead divided by the number of threads, which is 3 (Triple). 0.75 ÷ 3 = .250. The number of threads per inch (N) equals 1.0000 divided by the pitch. 1.0000 ÷ .250 = 4. The single depth equals .6866 divided by the number of threads per inch. .6866 ÷ 4 = .1716. The double depth equals .1716 multiplied by two, which is .3432.

Major Diameter - 3.0000
Double Depth - .3432
Minor Diameter - 2.6568

43. What kind of lubricant should be used for general threading?

A. Mineral lard oil is a very good lubricant for threading. It is made of white lead, graphite, and fatty oil. The Geometric Tool Company advises the use of the following compound for screw thread cutting.

40 gallons of water
10 gallons of mineral lard oil
2½ pounds of soda (exact quantity)

In some cases the addition of sulfur to a cutting compound proves very helpful.

44. How should the compound be set for cuting screw threads?

A. Do not force the tool into the work at 90 degrees with the center line, but swing the compound around one-half the included angle of the thread. For example, set the compound at an angle of 30 degrees for cutting American National threads, and at an angle of 14½ degrees for cutting Brown & Sharpe worm threads.

45. In setting the cutting tool to cut pipe threads, should the tool be set at 90 degrees with the center line of the work or at 90 degrees with the taper?

A. Set the tool at 90 degrees with the center line of the work.

46. Is the profile of the space the same as that of the thread? Give a reason for your answer.

A. Yes. A nut with a one inch major diameter and eight square threads per inch would not fit a one inch American National standard coarse thread screw which also had eight threads per inch.

47. What is the purpose of the pitch diameter?

A. The pitch diameter determines the fit of the thread.

48. Name the tools used in a Ford standard tap set.

A. Let us take a one-half inch tap set as an example.

½ inch body drill
13/32 tap drill
½ x 3/4 counterbore
½-13 std. taper tap
½-13 std. plug tap
½-13 std. bottoming tap

49. Name some methods of producing a thread.

A. Threads may be produced by revolving the work in the lathe against a proper cutting tool, by rolling the part to be threaded between dies in a special thread milling machine, by hand dies, and by cutting with a revolving cutter in a thread milling machine.

50. How is the size of the tap drill for a specified size determined?

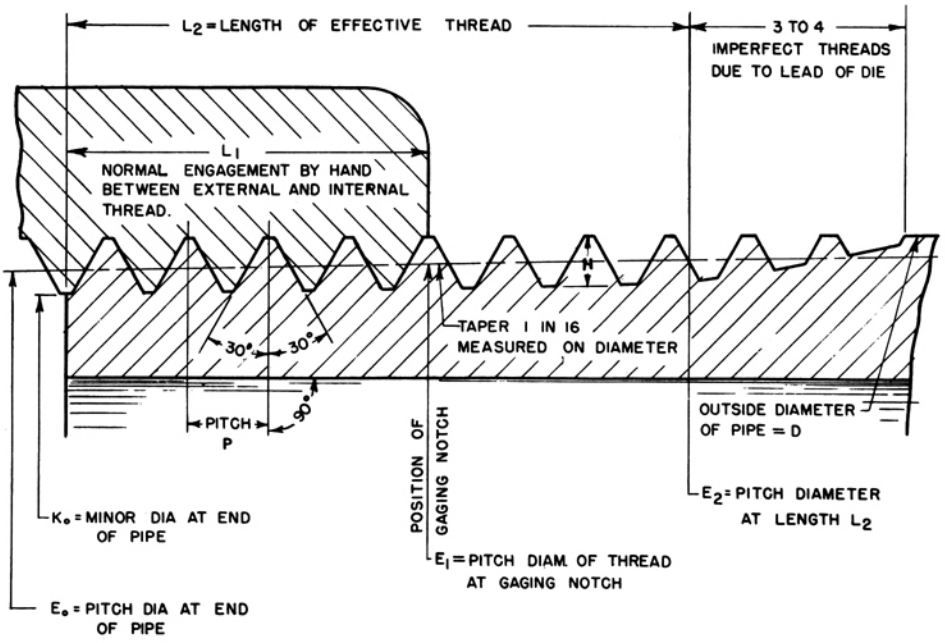
A. Divide 1.0825 by the number of threads per inch and subtract this result from the major diameter.

51. Why was the Acme thread designed?

A. Square threads were formerly used to a great extent on adjusting and power screws. The Acme thread was designed because of the difficulty of cutting square threads with taps and dies. It is much easier to produce accurately than the square thread. The Brown & Sharpe worm thread is deeper than the Acme thread.

52. Why are fine threads used on automobile parts instead of coarse threads?

A. Threads on automobile parts are cut in hard and tough materials, and need not be as coarse as threads cut in cast iron. A screw or bolt of a given size has a greater minor diameter and consequently a greater strength if the pitch is fine rather than coarse. A fine pitch screw or nut may be set tighter and does not shake loose as readily as one of a coarse pitch, although it must be remembered that a fine pitch thread will strip or shear easier than one of coarse pitch.


L2 = LENGTH OF EFFECTIVE THREAD
3 TO 4 IMPERFECT THREADS DUE TO LEAD OF DIE
L1
NORMAL ENGAGEMENT BY HAND BETWEEN EXTERNAL AND INTERNAL THREAD.
H
TAPER 1 IN 16 MEASURED ON DIAMETER
30°
30°
90°
PITCH P
POSITION OF GAGING NOTCH
OUTSIDE DIAMETER OF PIPE = D
E2 = PITCH DIAMETER AT LENGTH L2
K0 = MINOR DIA AT END OF PIPE
E1 = PITCH DIAM. OF THREAD AT GAGING NOTCH
E0 = PITCH DIA AT END OF PIPE

American National Taper Pipe Thread Notation

Notation

$E_0 = D - (0.005D + 1.1)p$
$E_1 = E_0 + 0.0625\ L_1$
$L_2 = p(0.8D + 6.8)$
$h = 0.8p.$

NATIONAL SCREW THREAD COMMISSION

Dimensions of national (American Briggs') taper pipe threads

Nominal size of pipe in inches	Number of threads per inch, n	Pitch, p	Depth of thread	Outside diameter of pipe, D	Length of normal engagement by hand, L_1	Length of effective thread, L_2	Increase in diameter per thread, $\frac{0.0625}{n}$	Pitch diameters			
								At end of pipe, or at length L_1 from end of coupling, $E_0 = D - \frac{0.05D+1.1}{n}$	At length L_1 on pipe, or at end of coupling, $E_1 = E_0 + \frac{L_1}{16}$		
								Basic	Maximum	Basic	Minimum
1	2	3	4	5	6	7	8	9	10	11	12
		Inch	*Inch*	*Inches*	*Inches*	*Inches*	*Inch*	*Inches*	*Inches*	*Inches*	*Inches*
⅛	27	0.03704	0.02963	0.405	0.180	0.26385	0.00231	0.36851	0.37833	0.37476	0.37129
¼	18	.05556	.04444	.540	.200	.40178	.00347	.47739	.49510	.48969	.48468
⅜	18	.05556	.04444	.675	.240	.40778	.00347	.61201	.63222	.62701	.62180
½	14	.07143	.05714	.840	.320	.55371	.00446	.75843	.78513	.77843	.77173
¾	14	.07143	.05714	1.050	.339	.54571	.00446	.96768	.99557	.98887	.98217
1	11½	.08696	.06957	1.315	.400	.66378	.00543	1.21363	1.24678	1.23863	1.23048
1¼	11½	.08696	.06957	1.660	.420	.70678	.00543	1.55713	1.59163	1.58338	1.57523
1½	11½	.08696	.06957	1.900	.420	.72348	.00543	1.79609	1.83049	1.82234	1.81419
2	11½	.08696	.06957	2.375	.436	.75652	.00543	2.26902	2.30442	2.29627	2.28812
2½	8	.12500	.10000	2.875	.682	1.18750	.00781	2.71963	2.77388	2.76316	2.75044
3	8	.12500	.10000	3.500	.766	1.20000	.00781	3.34062	3.40022	3.38950	3.37678
3½	8	.12500	.10000	4.000	.821	1.25000	.00781	3.83750	3.90053	3.88981	3.87709
4	8	.12500	.10000	4.500	.844	1.30000	.00781	4.33438	4.39884	4.38712	4.37540
4½	8	.12500	.10000	5.000	.875	1.35000	.00781	4.83125	4.89766	4.88594	4.87422
5	8	.12500	.10000	5.563	.937	1.40630	.00781	5.39073	5.46101	5.44929	5.43757
6	8	.12500	.10000	6.625	.956	1.51250	.00781	6.44609	6.51769	6.50597	6.49425
7	8	.12500	.10000	7.625	1.000	1.61250	.00781	7.43984	7.51406	7.50234	7.49062
8	8	.12500	.10000	8.625	1.063	1.71250	.00781	8.43359	8.51175	8.50003	8.48831
9	8	.12500	.10000	9.625	1.130	1.81250	.00781	9.42734	9.50969	9.49797	9.48625
10	8	.12500	.10000	10.750	1.210	1.93500	.00781	10.54531	10.63266	10.63094	10.60922
11	8	.12500	.10000	11.750	1.285	2.03500	.00781	11.53906	11.63110	11.61938	11.60766
12	8	.12500	.10000	12.750	1.360	2.12500	.00781	12.53281	12.63953	12.61781	12.60609
14 O. D.	8	.12500	.10000	14.000	1.562	2.25000	.00781	13.77500	13.88434	13.87262	13.86090
15 O. D.	8	.12500	.10000	15.000	1.667	2.35000	.00781	14.76875	14.88591	14.87419	14.86247
16 O. D.	8	.12500	.10000	16.000	1.812	2.45000	.00781	15.76260	15.88747	15.87575	15.86403
17 O. D.	8	.12500	.10000	17.000	1.900	2.55000	.00781	16.75625	16.88672	16.87500	16.86328
18 O. D.	8	.12500	.10000	18.000	2.000	2.65000	.00781	17.75000	17.88672	17.87500	17.86328
20 O. D.	8	.12500	.10000	20.000	2.125	2.85000	.00781	19.73750	19.88203	19.87031	19.85859
22 O. D.	8	.12500	.10000	22.000	2.250	3.06000	.00781	21.72500	21.87734	21.86562	21.85390
24 O. D.	8	.12500	.10000	24.000	2.375	3.26000	.00781	23.71250	23.87266	23.86094	23.84922
26 O. D.	8	.12500	.10000	26.000	2.500	3.46000	.00781	25.70000	25.86797	25.85625	25.84453
28 O. D.	8	.12500	.10000	28.000	2.625	3.66000	.00781	27.68750	27.86328	27.85156	27.83984
30 O. D.	8	.12500	.10000	30.000	2.750	3.86000	.00781	29.67500	29.85860	29.84688	29.83516

53. Where is the Buttress thread used?

A. The Buttress thread is used where a thread requiring great strength in one direction is required, as on vises, jacks, and power press screws.

54. What material is used between the threads of pipe and pipe coupling to prevent leakage?

A. Either red lead or white lead is generally used to seal the pipe couplings.

55. How can you distinguish a right-hand thread from a left-hand thread?

A. If the top of a bolt must be revolved to the right to be screwed into the receiving member, it is termed a right-hand thread, or if it must be revolved to the left, it is known as a left-hand thread.

56. Why is the O.D. of a 3/4 inch pipe much greater than 3/4 of an inch?

A. The inside diameter of a pipe is always as large or a little larger than the nominal size. For example, the inside diameter of a 3/4 inch pipe is .824 and the inside diameter of a one inch pipe is 1.049.

57. Describe the gaging of external and internal pipe threads.

A. In gaging external or male threads, the ring gage (Fig. 440), should screw tight by hand on the pipe or male thread until the small end of the gage is flush with the end of the thread.

In gaging internal or female threads, the plug gage (Fig. 441), should screw tight by hand into the fitting or coupling until the notch is flush with the face. When the thread is chamfered, the notch should be flush with the bottom of the chamfer.

Fig. 440. Ring Gage

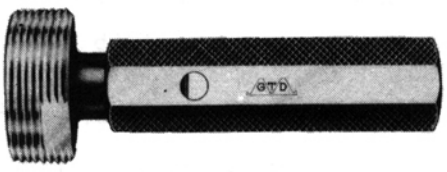


Fig. 441. Plug Gage

CAUTION. As the pipe tap is tapered, the hole is tapped for a short length only, before gaging with pipe tap gage. Never attempt to pass the pipe tap through the hole when tapping.

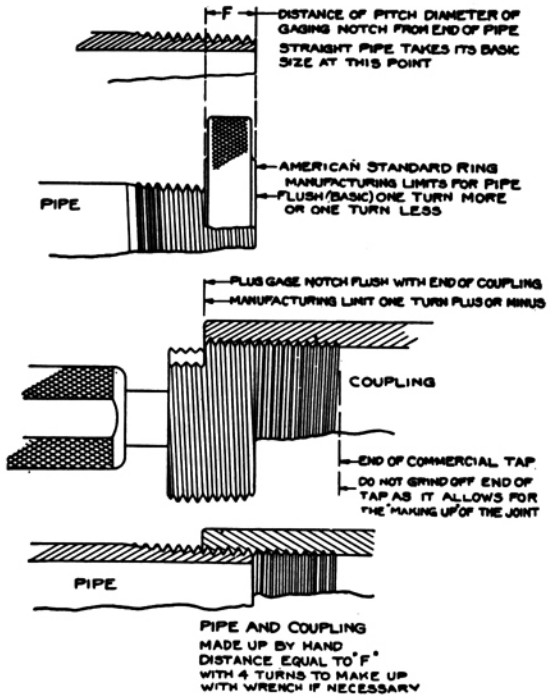

F
DISTANCE OF PITCH DIAMETER OF GAGING NOTCH FROM END OF PIPE
STRAIGHT PIPE TAKES ITS BASIC SIZE AT THIS POINT
AMERICAN STANDARD RING MANUFACTURING LIMITS FOR PIPE FLUSH (BASIC) ONE TURN MORE OR ONE TURN LESS
PIPE
PLUG GAGE NOTCH FLUSH WITH END OF COUPLING
MANUFACTURING LIMIT ONE TURN PLUS OR MINUS
COUPLING
END OF COMMERCIAL TAP
DO NOT GRIND OFF END OF TAP AS IT ALLOWS FOR THE MAKING UP OF THE JOINT
PIPE
PIPE AND COUPLING MADE UP BY HAND DISTANCE EQUAL TO "F" WITH 4 TURNS TO MAKE UP WITH WRENCH IF NECESSARY

Fig. 442. Use of Taper Pipe Gages

58. What measurement is checked with a thread micrometer?

A. The thread micrometer (Fig. 443) gages the thread on the pitch diameter.

Capacity	Range
1 in.	8 to 13 threads
1 "	14 " 20 "
1 "	22 " 30 "
1 "	32 " 40 "
2 "	4½ " 7 "
2 "	8 " 13 "
2 "	14 " 20 "
2 "	22 " 30 "

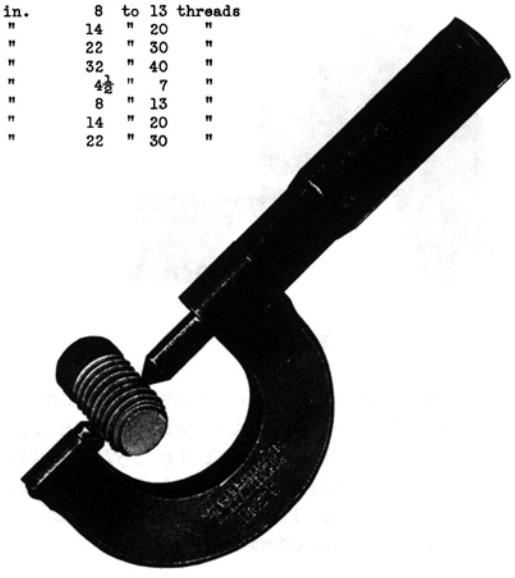


Fig. 443. Thread Micrometer

Capacity and Range of Thread Micrometers

Pitch Diameter = Major (Outside) Diameter—Depth of Thread

COARSE THREAD SERIES				FINE THREAD SERIES			
Size		Threads Per Inch	Cal. Reading or Pitch Diam.	Size		Threads Per Inch	Cal. Reading or Pitch Diam.
Number or Fraction	Decimal	N	$D-\frac{.649519}{N}$	Number or Fraction	Decimal	N	$D-\frac{.649519}{N}$
1	.0730	64	.0629	0	.0600	80	.0519
2	.0860	56	.0744	1	.0730	72	.0640
3	.0990	48	.0855	2	.0860	64	.0759
4	.1120	40	.0958	3	.0990	56	.0874
5	.1250	40	.1088	4	.1120	48	.0985
6	.1380	32	.1177	5	.1250	44	.1102
8	.1640	32	.1437	6	.1380	40	.1218
10	.1900	24	.1629	8	.1640	36	.1460
12	.2160	24	.1889	10	.1900	32	.1697
1-4	.2500	20	.2175	12	.2160	28	.1928
5-16	.3125	18	.2764	1-4	.2500	28	.2268
3-8	.3750	16	.3344	5-16	.3125	24	.2854
7-16	.4375	14	.3911	3-8	.3750	24	.3479
1-2	.5000	13	.4500	7-16	.4375	20	.4050
9-16	.5625	12	.5084	1-2	.5000	20	.4675
5-8	.6250	11	.5660	9-16	.5625	18	.5264
3-4	.7500	10	.6850	5-8	.6250	18	.5889
7-8	.8750	9	.8028	3-4	.7500	16	.7094
1	1.0000	8	.9188	7-8	.8750	14	.8286
1 1-8	1.1250	7	1.0322	1	1.0000	14	.9536
1 1-4	1.2500	7	1.1572	1 1-8	1.1250	12	1.0709
1 1-2	1.5000	6	1.3917	1 1-4	1.2500	12	1.1959
1 3-4	1.7500	5	1.6201	1 1-2	1.5000	12	1.4459
2	2.0000	4 1-2	1.8557	1 3-4	1.7500	12	†1.6959
2 1-4	2.2500	4 1-2	2.1057	2	2.0000	12	†1.9459
2 1-2	2.5000	4	2.3376	2 1-4	2.2500	12	†2.1959
2 3-4	2.7500	4	2.5876	2 1-2	2.5000	12	†2.4459
3	3.0000	*4	2.8376	2 3-4	2.7500	12	†2.6959
3	3.0000	3 1-2	2.8144	3	3.0000	10	†2.9350

59. When is the 3-wire system of checking the pitch diameter of screw threads used?

A. Three wires are laid in the thread groove and measured with an outside micrometer as shown in Figs. 444 and 445, when accurate pitch diameters on thread gages, taps, or pitch diameters beyond the range of available thread micrometers are checked.

Fig. 444

A set of wires consists of three wires having the same diameter within .00003", and a common diameter equal to .57735 divided by the number of threads per inch. Wires of this size will come in contact with the thread on the pitch diameter.

60. Find the micrometer reading over 3 wires (M) for a 3/4 N.C. thread.

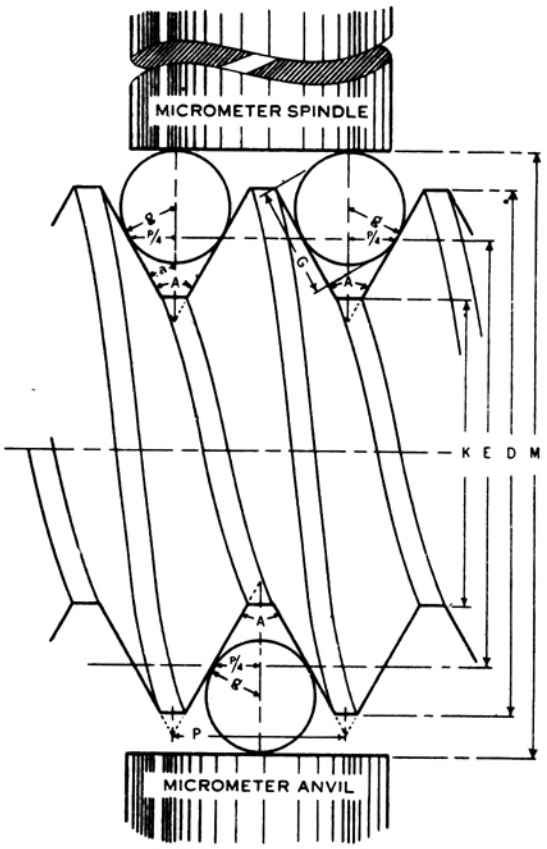


Fig. 445. Three-Wire System of Measuring Pitch Diameter.

M = measurement over wires
K = root diameter
A = angle of thread
D = major diameter of thread
G = diameter of wire
E = pitch diameter

A. Use the formula $M = D + 3G - \frac{1.5155}{N}$

To find the wire size to use, read down the column Threads per Inch to the required number of threads; then follow horizontally to the number in the column headed Diameter of Best Size Wires.

M = Measurement over the wires.
G = Diameter of the wire.
D = Major diameter of the thread.
N = Number of threads per inch.

$$M = D + 3G - \frac{1.5155}{N}$$

$$= .750 + 3 \times .05773 - \frac{1.5155}{10}$$

$$= .750 + .1732 - .15155$$

$$= .7717.$$

Threads per Inch	Pitch	Diameter of "Best-Size" Wires	Single Depth Nat. Form Thread	Width of Flat on Crest and Root NC and NF	Single Depth V-Thread	*
4	.250000	.1443375	.162379	.0312	.216506	.2344
4½	.222222	.1282998	.144337	.0278	.192449	.2187
5	.200000	.1154700	.129903	.0250	.173205	.1875
5½	.181818	.1049726	.118094	.0227	.157458	.1719
6	.166666	.0962246	.108253	.0208	.144336	.1562
7	.142857	.0824784	.092788	.0179	.123717	.1406
8	.125000	.0721687	.081189	.0156	.108253	.1250
9	.111111	.0641499	.072168	.0139	.096224	.1094
10	.100000	.0577350	.064951	.0125	.086602	.0938
11	.090909	.0524863	.059047	.0114	.078729	.0938
11½	.086956	.0502040	.056479	.0108	.075306	.....
12	.083333	.0481123	.054126	.0104	.072168	.0781
13	.076923	.0444114	.049963	.0096	.066617	.0781
14	.071428	.0412389	.046394	.0089	.061858	.0726
16	.062500	.0360843	.040594	.0078	.054126	.0625
18	.055555	.0320746	.036084	.0069	.048112	.0563
19	.052631	.0303865	.034185	.0065	.045579	.....
20	.050000	.0288675	.032475	.0062	.043301	.0469
22	.045454	.0262428	.029523	.0057	.039364	.0469
24	.041666	.0240558	.027063	.0052	.036083	.0422
27	.037037	.0213833	.024056	.0046	.032074	.0375
28	.035714	.0206194	.023197	.0045	.030929	.0360
30	.033333	.0192448	.021650	.0042	.028867	.0313
32	.031250	.0180421	.020297	.0039	.027063	.0313
34	.029411	.0169804	.019103	.0037	.025470	.....
36	.027777	.0160370	.018042	.0035	.024055	.....
40	.025000	.0144337	.016237	.0031	.021650	.....
44	.022727	.0131214	.014761	.0028	.019682	.....
48	.020833	.0120279	.013531	.0026	.018041	.....
50	.020000	.0115470	.012990	.0025	.017320	.....
56	.017857	.0103097	.011598	.0022	.015464	.....
64	.015625	.0090210	.010148	.0020	.013531	.....
72	.013888	.0080182	.009021	.0017	.012027	.....
80	.012500	.0072168	.008118	.0016	.010825	.....

*The figures in this column, when subtracted from basic major diameter, will give the nearest commercial standard drill size.

61. When the size of wire required to gage the thread on the pitch diameter cannot be obtained, state the method of finding the commercial size of wire that can be successfully used to check threads by the three-wire system.

A. When wires are used in conjunction with a micrometer for measuring screw threads, the minimum wire diameter must be such that the wires extend beyond the top of the screw to prevent the micrometer from bearing on the threads instead of on the wires, and the maximum limit must be such that the wires bear on the sides of the thread and not on the crest. The following formulas for determining the wire diameters do not give the extreme theoretical limits, but the smallest and largest sizes that are practicable.

The smallest diameter of wire to use is equal to .56 divided by the number of threads per inch.

The largest diameter of wire to use is equal to .90 divided by the number of threads per inch.

62. Describe the method of inspecting with a bench comparator.

A. Gaging with the bench comparator is a method of obtaining a very accurate comparison between a template and a job, Fig. 446 shows a Jones & Lamson standard bench comparator--inspecting a hob. A great variety of objects; such as hobs, taps, small tools, instrument gages, etc., can be inspected for accuracy by this method.

NUT THREADS

The difference in the size of the minor diameter of the nut and the minor diameter of the screw is called the clearance (see Fig. 447). The allowance for clearance is made in the nut.

$$\text{Clearance} = \frac{\text{Double Depth}}{6}$$

$$\text{Double Depth} = \text{Single Depth} \times 2 = \frac{1.299}{N}$$

$$\text{Single Depth} = \frac{.6495}{N}$$

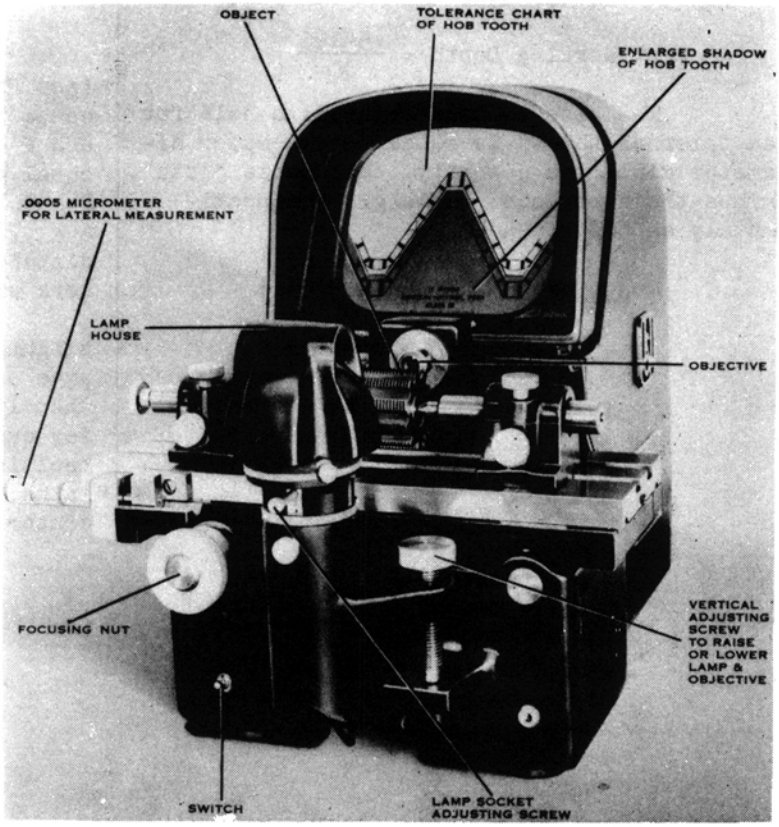


Fig. 446. Bench Comparator--Inspecting a Hob

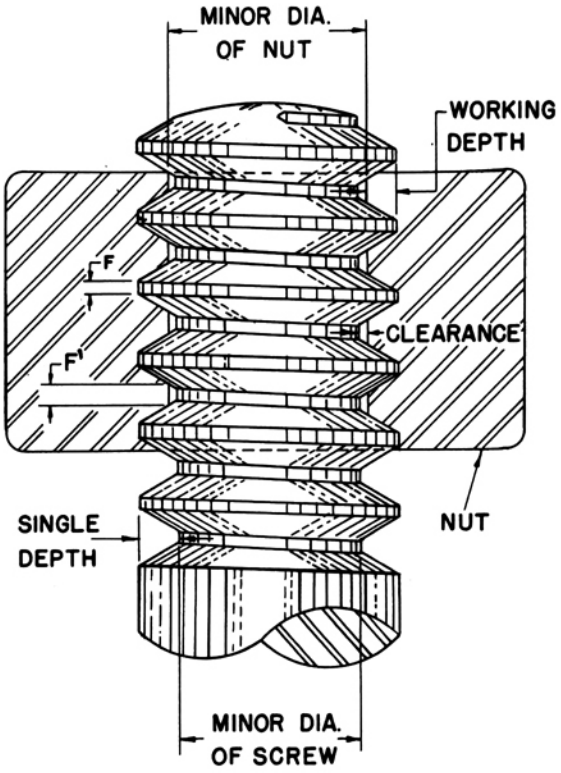


Fig. 447

$$\text{Working Depth} = \frac{.54125}{N}$$

The size to bore or drill a hole for an internal thread is equal to the major diameter minus the quantity, the double depth minus the clearance. This gives approximately an 80% thread.

$$\text{Major Dia.} - \left(\frac{1.299}{N} - \frac{1.299}{6N}\right)$$

$$= \text{Major Dia.} - \frac{1.0825}{N}$$

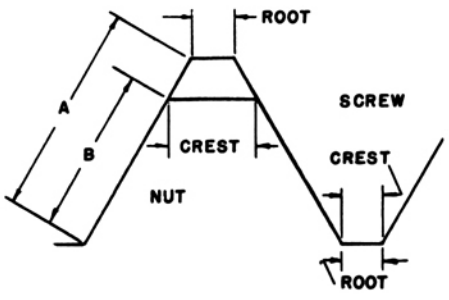


Fig. 448

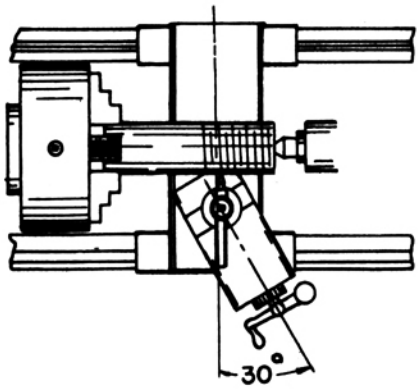


Fig. 449. External Threading

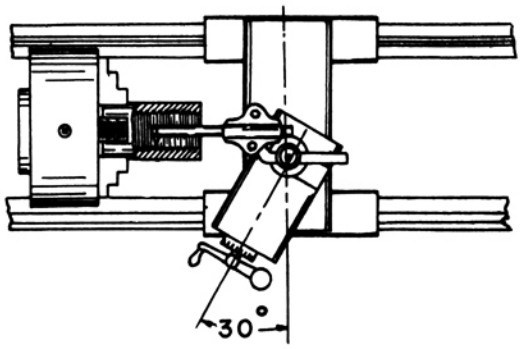


Fig. 450. Internal Threading

The crest of the thread in the nut (see Fig. 448) is larger than the root because the clearance is cut in the nut (see F and F' in Fig. 447). When the clearance is equal to DD ÷ 6, the crest F'is equal to P ÷ 4. When the compound is set at 30°, A = .750 ÷ N and B = .625 ÷ N. This is the distance to force the threading tool into the work with the compound feed.

In practice the compound is set at slightly less than 30°, so that there is clearance on the back edge of the threading tool when it is fed in by the compound. The last few cuts are taken by feeding the threading tool into the work with the cross feed of the lathe. The threading tool must be set on center (see page 140, Lathe Section).

Chapter 15

GEARING

Spur Gearing

A gear is a toothed wheel used to transmit positive and uniform rotary motion from one shaft to another.

Gears are divided into groups and named according to the position which the teeth occupy with respect to the axis of rotation of the gear body. Among the more common types of gears in use we find the SPUR GEAR, BEVEL GEAR, HELICAL GEAR, WORM and WORM GEAR, HERRINGBONE GEAR, and SPIRAL GEAR. Some special gears are used, such as square gears, elliptical gears, planetary gears, and intermittant gears. These special gears all have a definite purpose and are suitable for certain conditions, but are not used nearly as much as the first mentioned group.

SPUR GEARS, Fig. 451, are gears having straight teeth cut parallel with the

Fig. 451

axis of rotation of the gear body. They are used to connect shafts whose axis are parallel.

BEVEL GEARS, Fig. 452, are gears having teeth cut so that they radiate from the apex of a cone and lie on the conical surface. They are used to transmit motion

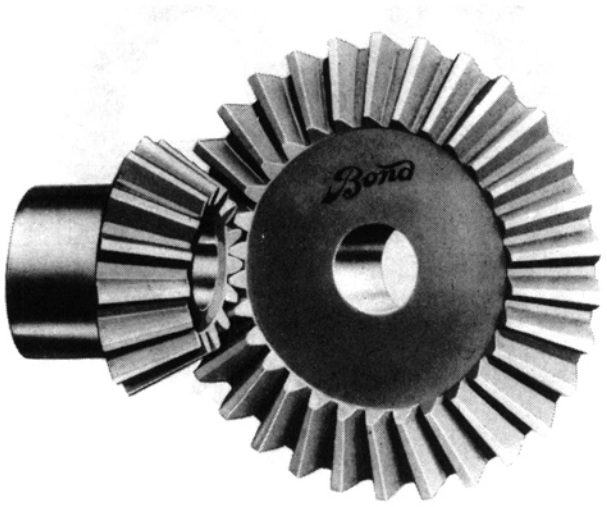


Fig. 452

between shafts whose center lines intersect. Miter gears are mating bevel gears having the same number of teeth and whose pitch cone angles are 45°.

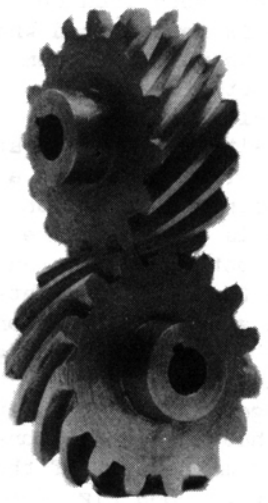


Fig. 453

HELICAL GEARS, Fig. 453, are gears having teeth cut on a cylinder and at an angle with the axis of rotation of the gear body. These gears are sometimes called spiral gears by mistake. They are used in machines where the shafts do not intersect and where quiet, smooth operation is required. Since the contact action of these gears is sliding rather than a rolling action, they should be run in an oil bath.

A WORM GEAR is a gear having the teeth cut angular with the axis of rotation of the gear body and radially in the gear

face. A worm gear is driven by a WORM which resembles a large screw thread. These gears give very quiet operation and have a long life. They are used extensively as speed reducers. Fig. 454 illustrates a worm and a worm gear.

Fig. 454

HERRINGBONE GEARS, Fig. 455, are gears having helical teeth which diverge from the center of the face toward the sides of the gear body. They have a particular advantage over the other types of gears, in that tooth load or shock is neutralized when it is transmitted from one tooth to another, thus eliminating the necessity of thrust bearings. They also give a more continuous action which is entirely free from tooth vibration such as is found in straight-tooth gears. Herringbone gears are used where high speeds and high gear ratios are necessary.

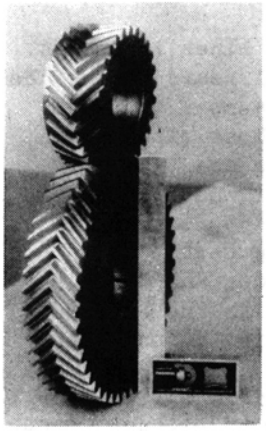


Fig. 455

SPIRAL GEARS are gears with teeth cut on a conical surface so that they curve continually toward or away from the apex of the cone upon which they are cut. See Fig. 456. These gears closely resemble bevel gears and are frequently called spiral bevel gears. This type of gear can be used satisfactorily for ratios as high as 6 to 1 and give quiet, efficient operation. They are used extensively in the rear axles of automobiles.

Fig. 456

In the formation of gear teeth, two systems are in general use, the INVOLUTE and the INVOLUTE STUB. The involute tooth has a pressure angle of $14\frac{1}{2}^{\circ}$ and is the form most commonly used. The involute stub, commonly known as the STUB TOOTH, has a standard pressure angle of 20°; however, some deviations are made from this standard value. The stub tooth has two pitch sizes given in the form of a fraction. The numerator of the fraction indicates the diametral pitch to be used in calculating the chordal thickness and the pitch diameter. The denominator of the fraction is the diametral pitch to be used in calculating the addendum, dedendum, and whole depth.

The pressure angle is, as the name implies, the angle at which the tooth pressure is applied and distributed. Through experience it was found that for maximum efficiency the pressure angle should be $14\frac{1}{2}^{\circ}$. As this angle of pressure becomes greater the outward strain upon the shaft bearings becomes greater, which increases the friction on the bearings with a consequent shortening of bearing life. It can be readily understood that, where a stub tooth gear is desirable for longer tooth life, special bearings must be installed to compensate for the added friction and to maintain a correct center distance. In Fig. 457 "Z" indicates the pressure angle.

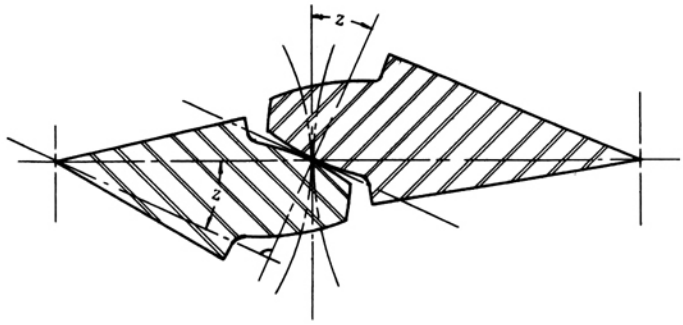


Fig. 457

In order to calculate gearing properly, a thorough knowledge of the terms used is necessary. For example, one should know exactly what the term PITCH means with respect to gears. In this case pitch means frequency. The DIAMETRAL PITCH means the frequency with which the teeth occur in 3.1416 inches measured on the pitch circumference. It should be remembered that the term DIAMETRAL PITCH means a quantity; whereas PITCH DIAMETER is a distance which depends on the value of the diametral pitch. If any of these terms are confused it may result in the use of the wrong formula; consequently the calculations will be in error. When the CENTER DISTANCE is spoken of, one should think of two circles either externally or internally tangent to each other. The distance between the centers of these circles is the CENTER DISTANCE. This dimension is very important, because the gears must operate freely and quietly at exactly this distance.

The following definitions cover the terms most frequently used in spur gearing:

The PITCH CIRCLE of a gear is an imaginary circle located about half-way down the teeth, where the teeth of both gears contact each other.

The PITCH DIAMETER is the diameter of the pitch circle, and is important because it regulates the pitch circle or the point of contact between two gears.

The DIAMETRAL PITCH is the number of teeth in 3.1416 inches measured on the pitch circle, or the number of teeth for each inch of pitch diameter. It regulates the size of the tooth.

The CIRCULAR PITCH is the distance from the center of one tooth to the center of the next consecutive tooth measured on the pitch circle.

The ADDENDUM is that portion of the tooth which projects above or outside of the pitch circle.

The DEDENDUM is that portion of the tooth between the pitch circle and the root circle and is equal to one addendum plus the clearance.

The ROOT CIRCLE is the circle formed by the bottoms of the teeth.

The WHOLE DEPTH is the distance from the top of the tooth to the bottom and consists of two addendums plus the clearance.

The BASE CIRCLE is, as the name implies, the basic circle upon which the teeth are mechanically constructed or drawn in the drafting room. It also forms an important part in calculating the pressure angle. To find the radius of the base circle multiply the pitch radius by the Cosine of the pressure angle.

It must be remembered that the diametral pitch regulates the size of the tooth much the same as the number of threads per inch regulates the size of a screw thread. The diametral pitch is always given in the form of a number as, 2, 3, 4, 8, etc.; and as this number gets larger the size of the tooth becomes smaller. For example a 2 diametral pitch gear would have a much larger tooth than a 10 diametral pitch gear, because in the case of the 10 diametral pitch more teeth are being crowded into 3.1416 inches. (TWO GEARS MUST BE OF THE SAME DIAMETRAL PITCH BEFORE THEY CAN MESH) For example two 7 diametral pitch spur gears will mesh, but a 7 diametral pitch gear will not mesh with a 10 diametral pitch gear.

When motion is transmitted from one shaft to another through the medium of gear teeth, it is of the utmost importance that they be actually as well as theoretically correct. To check the size of a gear tooth to any degree of accuracy, it must be measured on a straight line from where the pitch circle touches the tooth (a) on one side (see Fig. 458) to where it leaves the tooth

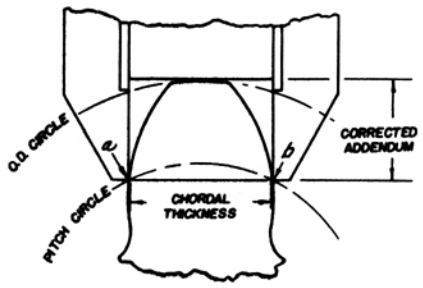


Fig. 458

on the other side (b). This thickness is known as the CHORDAL THICKNESS, because geometrically it is the length of a chord. In order to measure the true length of this chord the corrected addendum must be used.

The CORRECTED ADDENDUM is the perpendicular distance from the chord at the pitch circle to the top of the gear tooth. Why this corrected addendum must be used can be readily understood from the sketch in Fig. 458. A calculated distance equal to the height of the arc above the chord when added to "S" gives the distance "H". When the addendum beam on the gear tooth vernier caliper is set at this distance (H), it permits the jaws to contact the gear tooth exactly at the pitch line. The height of the

arc above the chord for the different diametral pitches and for varying numbers of teeth have been calculated and charted. It is convenient to use the values from these charts when checking gear tooth parts.

To test the thickness of a gear tooth with a gear tooth vernier caliper, the addendum beam is set for the corrected addendum. Then, after the burrs have been stoned from the face of the tooth, the addendum beam is set on top of the tooth as shown in Fig. 459. The caliper jaws are adjusted to the tooth thickness by means of the knurled nut "A". The chordal thickness can then be read from the "B" scale.

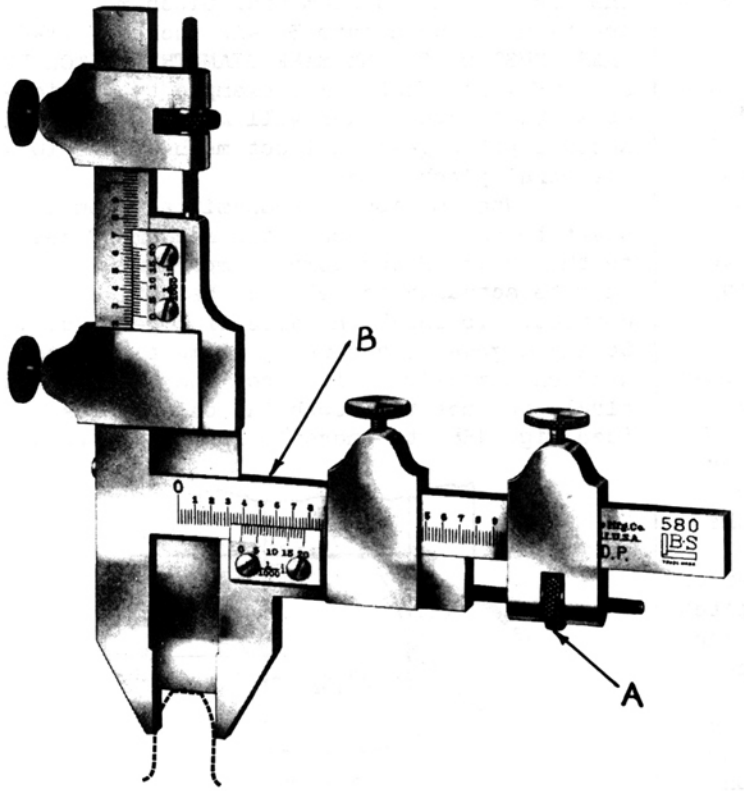


Fig. 459

CAUTION: Make certain that the vernier graduations are read correctly. The smaller sizes of calipers have graduations equal to .020 of an inch, while the larger sizes have graduations equal to .025 of an inch.

It must be remembered that this tool checks the corrected addendum and chordal thickness but does not check the pitch diameter of a gear. To check the pitch diameter of a gear accurately the diameters of plug gages are calculated so that they will be tangent to the teeth at the pitch circle. Then by means of trigonometry the distance over the outside of these plugs is calculated and measured with a micrometer.

The following notations and abbreviations will be used in the formulas for spur gear calculations:

Ng = Number of teeth in the gear
Np = Number of teeth in pinion
P = Diametral Pitch
P' = Linear or Circular Pitch
D = Pitch Diameter
O = Outside Diameter
H = Corrected Addendum
I = Inside Diameter
S = Addendum
S + F = Dedendum
F = Clearance
W = Whole Depth
C = Center Distance
T = Chordal Thickness
L = Length of Rack
Z = Pressure Angle

The following rules and formulas will be used in calculating the dimensions for spur gears:

To Find	Formula
Diametral Pitch	$P = \frac{3.1416}{P'}$
Diametral Pitch	$P = \frac{Ng + Np}{2C}$
Diametral Pitch	$P = \frac{N + 2}{O}$
Diametral Pitch	$P = \frac{N}{D}$
Circular Pitch	$P' = \frac{3.1416}{P}$
Pitch Diameter	$D = \frac{N}{P}$
Pitch Diameter	$D = O - 2S$
Outside Diameter	$O = \frac{N + 2}{P}$
Outside Diameter	$O = D + 2S$
Number of Teeth	$N = P \times D$
Addendum	$S = \frac{1}{P}$
Addendum	$S = \frac{P'}{3.1416}$
Dedendum	$S + F = \frac{1.157}{P}$
Clearance	$F = \frac{.157}{P}$
Clearance	$F = \frac{3.1416}{20 \times P}$

To Find	Formula
Whole Depth	$W = \frac{2.157}{P}$
Whole Depth	$W = .6866 \times P'$
Center Distance	$C = \frac{Ng + Np}{2P}$
Length of Rack	$L = NP'$

For internal spur gears use the following formulas:

To Find	Formula
Inside Diameter of Internal Gears	$I = \frac{N - 2}{P}$
Center Distance of Internal Gears	$C = \frac{Ng - Np}{2P}$
Diametral Pitch of Internal Gears	$P = \frac{Ng - Np}{2C}$

In the following gear tooth parts chart, "N" is the number of teeth in the gear to be cut, "T" is the chordal thickness, and "H" is the corrected addendum of a ONE diametral pitch gear.

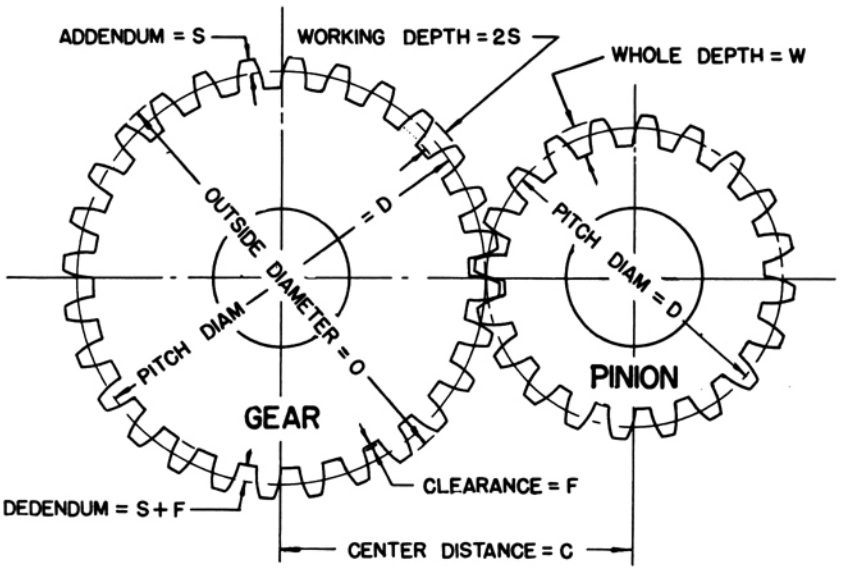
N	T	H
8	1.5607	1.0769
9	1.5628	1.0648
10	1.5643	1.0616
11	1.5654	1.0559
12	1.5663	1.0514
14	1.5675	1.0444
17	1.5686	1.0362
21	1.5694	1.0294
26	1.5698	1.0237
35	1.5702	1.0176
55	1.5706	1.0112
135	1.5707	1.0047

RACK TOOTH

The Diametral Pitch (Pitch or P.) indicates the size of the tooth.

$$\text{Addendum} = \frac{1}{P} \qquad \text{Dedendum} = \frac{1.157}{P}$$

$$\text{Linear Pitch} = \frac{\pi}{P}$$



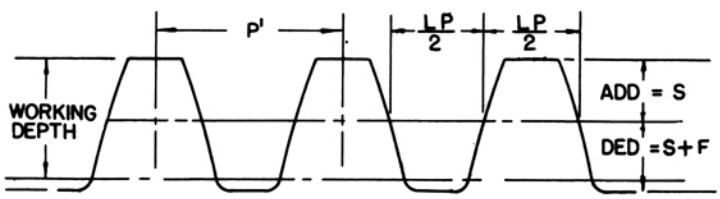


Fig. 460

Working Depth = $\frac{2}{P}$ Whole Depth = $\frac{2.157}{P}$

Linear or Circular Pitch = P'

To find the chordal thickness or corrected addendum of a gear, divide the number under "T" or "H", corresponding to the nearest number of teeth in the gear, by the diametral pitch. Example: Find the chordal thickness and corrected addendum for a gear having 74 teeth and a diametral pitch of 8. Referring to the above chart, find the number of teeth nearest to 74, which is 55. Opposite 55 under "T" find the value 1.5708, which is the chordal thickness for a ONE diametral pitch gear. Divide 1.5708 by 8 to get .1963, which is the chordal thickness required. Opposite 55 under "H" find 1.0112 and divide it by 8 to get .1264, which is the corrected addendum required. Therefore an 8 diametral pitch gear having 74 teeth will have a chordal thickness of .1963 and a corrected addendum of .1264.

In using a milling machine for cutting the teeth in a gear, it is necessary to use a rotary gear cutter. This rotary cutter must be selected so that it will give the gear teeth the correct curvature. The cutters are selected according to numbers, depending on the number of teeth in the gear to be cut and the diametral pitch. The cutter number merely represents the curvature on the sides of the teeth. Therefore when milling the teeth in a gear with a rotary milling cutter it is necessary to give the cutter number on the tool crib requisition.

The following chart gives the numbers of rotary cutters which will cut from 12 teeth up to and including a rack:

No. 1	for	135	teeth	to a	rack
" 2	"	55	"	"	134 teeth
" 3	"	35	"	"	54 "
" 4	"	26	"	"	34 "
" 5	"	21	"	"	25 "
" 6	"	17	"	"	20 "
" 7	"	14	"	"	16 "
No. 8	for	12	teeth	to	13 teeth

The following problems illustrate the use of the notations and formulas given on pages 104 and 105.

PROBLEM:

Two gears, A and B, have 48 and 27 teeth respectively and are of 12 diametral pitch. In gear A find 1) the outside diameter, 2) the chordal thickness, 3) the corrected addendum. In gear B find 4) the circular pitch, 5) the pitch diameter, 6) the number of the rotary gear cutter used for milling the teeth.

Procedure:

According to the formulas:

1. $O = \frac{N + 2}{P}$; $O = \frac{48 + 2}{12}$; $O = \frac{50}{12}$;

   $O = 4.1666$.

2. In solving for the chordal thickness, "T", refer to the table of gear tooth parts on page 105. The given number of teeth for which the chordal thickness must be found is 48; the nearest number of teeth to 48 is 55. Taking the value for "T" opposite 55 and dividing by the diametral pitch, 12, gives .1309 as the chordal thickness of a 12 pitch gear having 48 teeth.

3. To solve for the corrected addendum refer to the table of gear tooth parts on page 105. The number of teeth nearest to 48 is 55; opposite 55, under "H" find the value 1.0112. Divide this value by the diametral pitch, 12, which gives .0842 as the corrected addendum of a 12 diametral pitch gear having 48 teeth.

4. $P' = \frac{3.1416}{P}$; $P' = \frac{3.1416}{12}$; $P' = .2618$

5. $D = \frac{N}{P}$; $D = \frac{27}{12}$; $D = 2.25$

6. Referring to chart, find what number corresponds to a 26 tooth gear. In this case it is found that a #4 cutter will cut from 26 to 34 teeth; therefore a #4 rotary cutter of 12 pitch should be taken from the crib to cut the number of teeth in the pinion.

PROBLEM:

A gear has 53 teeth and a pitch diameter of 3.7857". Calculate the following: 1) Find P; 2) find O; 3) find W; 4) find T; 5) find P'; 6) find S; 7) find H; 8) find F; 9) find S + F; 10) What number of cutter should be used for cutting this gear?

Procedure:

1) $D = \frac{N}{P}$; $3.7857 = \frac{53}{P}$; $3.7857\ P = 53$;

   $P = \frac{53}{3.7857}$; $P = 14$

2) $O = \frac{N + 2}{P}$; $O = \frac{53 + 2}{14}$; $O = \frac{55}{14}$;

   $O = 3.9285$

3) $W = \frac{2.157}{P}$; $W = \frac{2.157}{14}$; $W = .15407$

4) $T = \frac{1.5708}{P}$; $T = \frac{1.5708}{14}$; $T = .11213$

5) $P' = \frac{3.1416}{P}$; $P' = \frac{3.1416}{14}$; $P' = .2244$

6) $S = \frac{1}{P}$; $S = \frac{1}{14}$; $S = .0714$

7) $H = \frac{1.011}{P}$; $H = \frac{1.011}{14}$; $H = .0722$

8) $F = \frac{.157}{P}$; $F = \frac{.157}{14}$; $F = .0112$

9) $S + F = \frac{1.157}{P}$; $S + F = \frac{1.157}{14}$;

$S + F = .08264$

10) For 53 teeth select a #3 cutter.

PROBLEM:

Two gears, A and B, are of 9 diametral pitch. Gear A has an outside diameter of 8.1111". Gear B has 38 teeth. Calculate (1) the number of teeth for gear A, (2) the center distance, (3) the circular pitch, (4) the whole depth, (5) the pitch diameter of gear A, (6) the outside diameter of gear B, (7) the dedendum, (8) the pitch diameter of gear B, (9) the cutter number for gear A, (10) the cutter number for gear B.

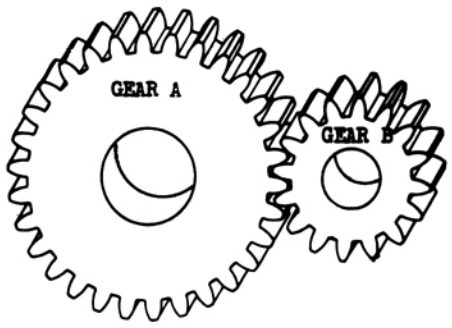


Fig. 461

Procedure:

1) $O = \frac{N + 2}{P}$; $8.1111 = \frac{N + 2}{9}$; $N + 2$

$= 72.9999$; $N = 72.9999 - 2$ or $N = 71$ teeth

2) $C = \frac{Ng + Np}{2P}$; $C = \frac{71 + 38}{2 \times 9}$; $C = \frac{109}{18}$;

$C = 6.0555$

3) $P' = \frac{3.1416}{P}$; $P' = \frac{3.1416}{9}$; $P' = .34906$

4) $W = \frac{2.157}{P}$; $W = \frac{2.157}{9}$; $W = .23966$

5) $D = \frac{Ng}{P}$; $D = \frac{71}{9}$; $D = 7.8888$

6) $O = \frac{Np + 2}{P}$; $O = \frac{38 + 2}{9}$; $O = 4.4444$

7) $S + F = \frac{1.157}{P}$; $S + F = \frac{1.157}{9}$;

$S + F = .12855$

8) $D = \frac{Np}{P}$; $D = \frac{38}{9}$; $D = 4.2222$

9) For 71 teeth select a #2 cutter.

10) For 38 teeth select a #3 cutter.

The following problems are typical of those encountered in the shop and drafting rooms and should be worked out very carefully:

1. Make a working drawing of each gear of a pair of spur gears, having a 2 to 1 ratio and a 3" center distance. Choose bore and pitch to suit.

2. Make a working drawing of each gear of a pair of spur gears, having a 2 to 1 ratio and a 2 17/32" center distance. Choose bore and pitch to suit.

3. A rack measures approximately 29 1/32" over 37 teeth. Calculate all dimensions necessary for drawing a gear that will move the rack 25 1/8" when the gear makes one revolution.

4. A spur gear having a 1" face, a 2.083" outside diameter, and 23 teeth is mounted on a pump shaft which is run at 300 R.P.M. This gear is driving a second gear running at 100 R.P.M. Calculate the dimensions necessary for both gears.

5. Calculate the dimensions necessary to make drawings of each gear of a pair of stub tooth gears, 4-6 pitch, 2 to 1 ratio, if the center distance is 4.5".

6. A gear revolving through an angle of $112\frac{1}{2}^{\circ}$ causes a rack to move 4 11/64". Calculate the dimensions necessary for drawing the gear.

7. A gear having 75 teeth and an outside diameter of 9.625" runs at 100 R.P.M. The pinion runs at 312.5 R.P.M. Calculate all the dimensions for both gears.

8. What diametral pitch is equivalent to .52432" circular pitch?

9. What number of teeth should a gear contain if the outside diameter is 18" and the diametral pitch is 8?

10. What is the outside diameter of a gear having 70 teeth and an 8 diametral pitch?

11. Find the chordal thickness and corrected addendum of a gear having 36 teeth and 5 diametral pitch.

12. The center line of a shaft running at 700 R.P.M. is located $6\frac{1}{2}$" from the center of another shaft that is to run at 300 R.P.M. Make a detail drawing of each gear of a pair of spur gears to connect shafts. Find necessary dimensions.

13. What is the diametral pitch of a gear if the circular pitch is .7854?

14. How many teeth should there be in a spur gear if the outside diameter is 18" and the diametral pitch is 6?

15. What is the outside diameter of an 8 diametral pitch spur gear having 70 teeth?

16. Two 8 diametral pitch spur gears have 24 and 72 teeth respectively, what is their center distance?

17. The ratio of two spur gears is 4 to 5 and the center distance is 27". What number of teeth does each gear have if the diametral pitch is 5?

18. The speed ratio of two spur gears is 3 to 4, the diametral pitch is 10 and the center distance is 21". Find the outside diameter, the pitch diameter, and the number of teeth in each gear.

19. Find the pitch of a spur gear which has 80 teeth and an outside diameter of 16.400".

20. If a spur gear has 140 teeth and an outside diameter of 10 1/7", what is its diametral pitch?

Figures 462 to 465 show some of the methods of producing gears.

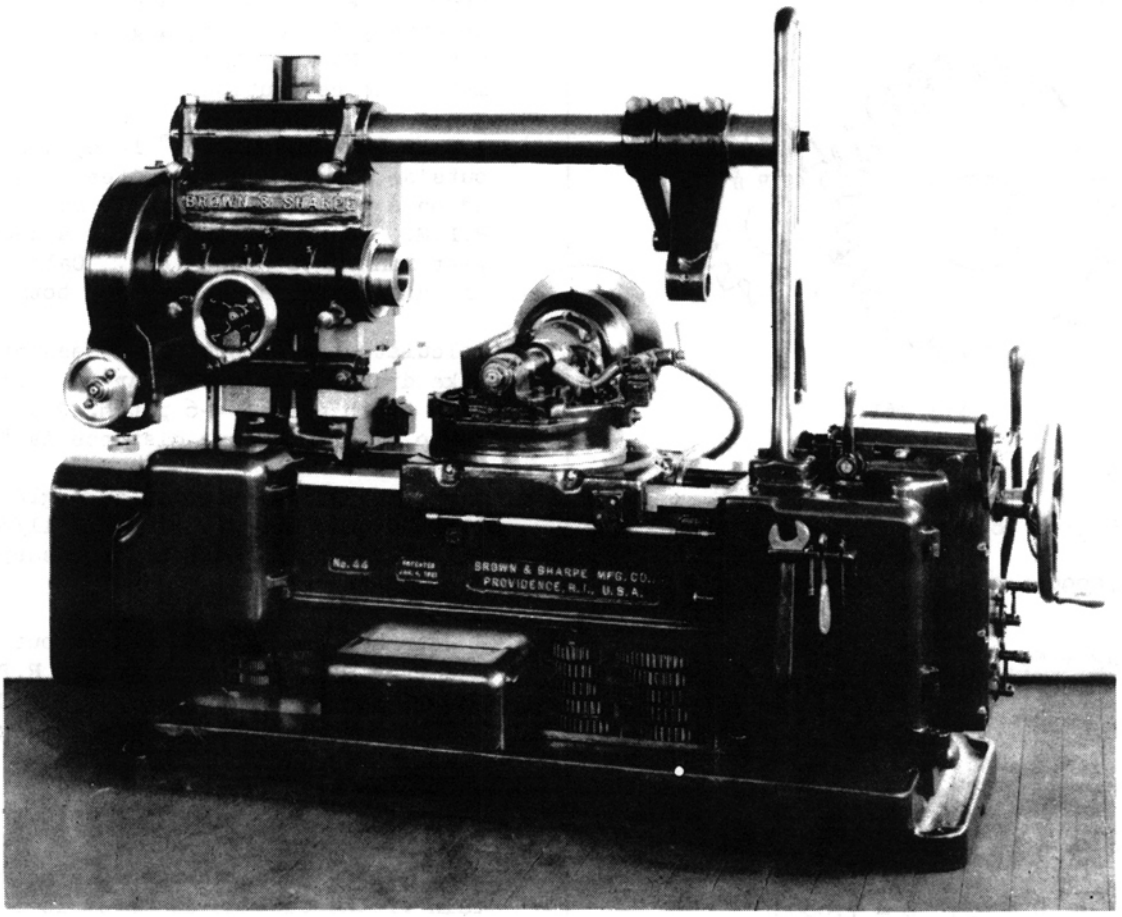


Fig. 462. Gear Hobbing Machine for Spur and Helical Gears.

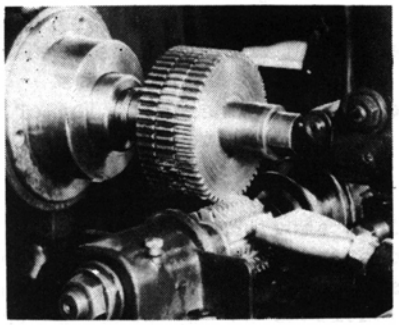


Fig. 463. Hobbing Spur Gears.

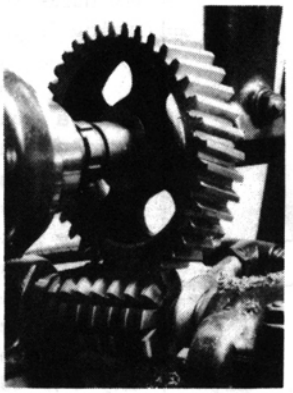


Fig. 464. Hobbing Helical Gears.

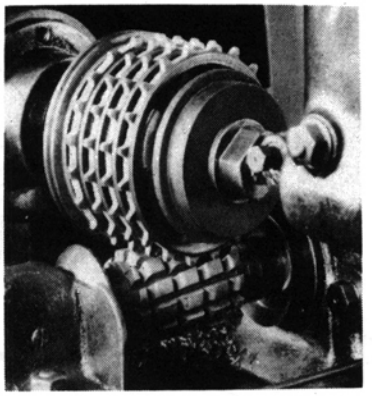


Fig. 465. Hobbing Sprockets

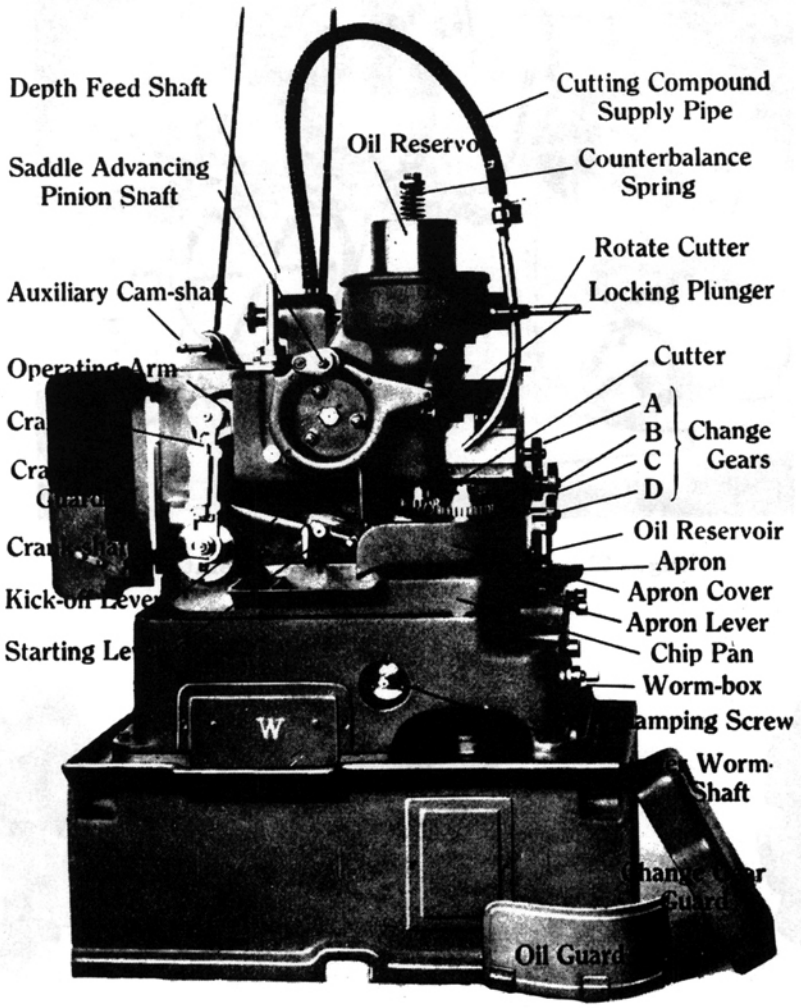


Fig. 466. Front View of Fellows Gear Shaper with All Guards Removed.

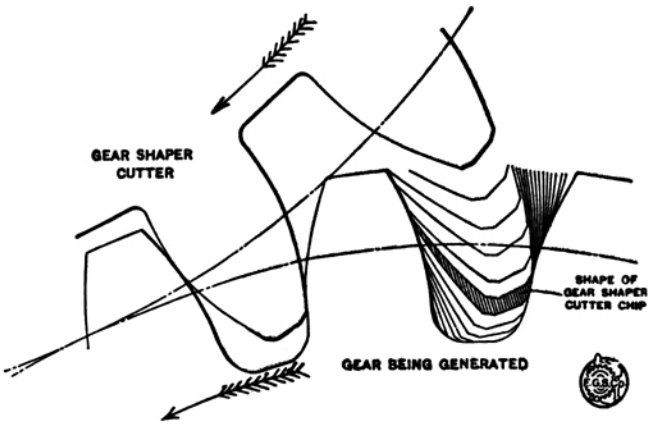


Fig. 467
Shows the action of the cutter and work.

Fig. 466 shows the principal operating members of a Fellows Gear Shaper. This machine is designed to cut one gear at a time using high reciprocating cutter speeds combined with a short cutting stroke.

The Gear Shaper Cutter, Fig. 467, can take a coarser feed than the milling machine type of cutter. The cutter and work are shown rotating together in the direction of the arrows. The cutter is given a reciprocating motion at right angles to the illustration, similar to that of a planning tool.

The outlines show the various positions which the cutting edge will occupy for each successive stroke. The distance between any two adjacent outlines at any point is the thickness of the chip at that point.

Fig. 468 shows the Gleason Bevel Gear Generator used for machining the teeth on bevel gears, a two-tool machine for rough and

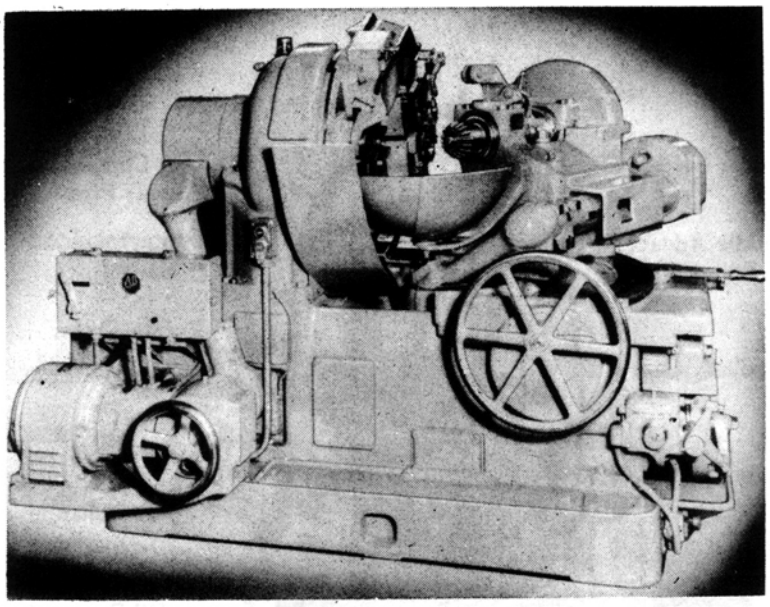


Fig. 468.. 12" Straight Bevel Gear Generator.

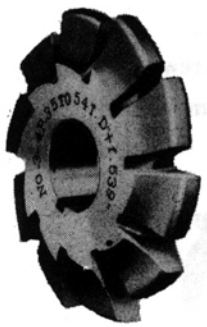


Fig. 469

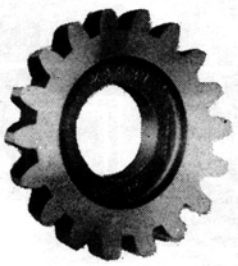


Fig. 470

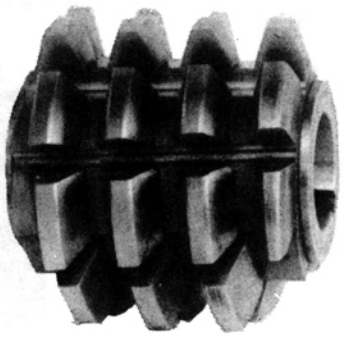


Fig. 471

finish cutting.

Fig. 469 shows an Involute Form cutter used on a milling machine for cutting gear teeth. Fig. 470 shows the kind of cutter used for producing gear teeth by the gear shaper method, while in Fig. 471 is a Hob, used for producing gear teeth by the hobbing method.

Gearing Questions

1. What is a gear?
2. From what factors do gears derive their names?
3. Name six kinds of gears that are commonly used.
4. Name three kinds of special gears.
5. What is a spur gear?
6. What are bevel gears? For what are they used?
7. Are all bevel gears miter gears? Explain.
8. What is the difference between a spiral gear and a helical gear?
9. What advantages do helical and spiral gears have over spur and bevel gears?
10. What is a worm gear?
11. What is the purpose of a worm and worm gear?
12. What is a herringbone gear?
13. What particular advantages do herringbone gears possess?
14. When is it advisable to use herringbone gears?
15. What are spiral gears?
16. How do spiral gears differ from helical gears?
17. Where are spiral gears generally used?
18. What is meant by the involute form of gear tooth?
19. Why are involute gear teeth used most?
20. What is a stub tooth gear?
21. How is the pitch of a stub tooth gear generally specified?
22. Under what conditions is it advisable to use a stub tooth gear?
23. What are the disadvantages of the stub tooth gear?
24. What is meant by the pressure angle of a gear?
25. Why is the pressure angle important?
26. What is meant by the pitch of a gear?
27. Why is it necessary to have a good understanding of the terms used in gearing?
28. What is the difference between diametral pitch and circular pitch?
29. Could gears of different diametral pitches be made to run with each other? Why?
30. Why is the pitch diameter of a gear important?
31. How does the length of the addendum affect the chordal thickness?
32. Under what conditions would it be necessary to know the diameter of the base circle?
33. Why is the center distance important?
34. How are diametral and circular pitch sizes generally specified?
35. Could a 4 diametral pitch gear by made to run with an 8 diametral pitch gear? Why?
36. What is the difference between the addendum and the corrected addendum?
37. Explain how to check the chordal thickness of a gear tooth with a gear tooth vernier caliper.
38. Under what conditions would it be possible to have the chordal thickness correct and yet not have the gears run together properly?
39. For precision gear work how should the gear be checked?
40. How is the chordal thickness determined for a gear having a definite number of teeth and diametral pitch?
41. How is the corrected addendum determined for a gear having a definite number of teeth and diametral pitch?
42. In using the milling machine for cutting the teeth in a gear what kind of a cutter is used?
43. What two things govern the selection of the cutter?
44. Why is it necessary to have a great many cutters for milling the teeth in gears of the same diametral pitch?
45. Give three methods of cutting gears.

Chapter 16

CUTTING TOOLS

The successful operation of a lathe, shaper, planer, etc., turning out work of good quality, depends to a great extent on the skill of the operator in grinding his tools. Dull and improperly ground tools throw a heavy strain on the feed mechanism, causing the work to spring and the machine to chatter.

Cutting tools are made of carbon steel, high speed steel, and alloys such as stellite and tungsten carbide. The stellite and tungsten carbide tools are used only on high production work because their cost is too great for universal use. Some carbon steel tools are still used but the general practice is to use high speed steel tool bits in holders. A quick and simple method to tell whether a tool is carbon or high speed steel is to grind the end and watch the sparks. The wheel will throw light-colored sparks on carbon steel and dark red sparks on high speed steel.

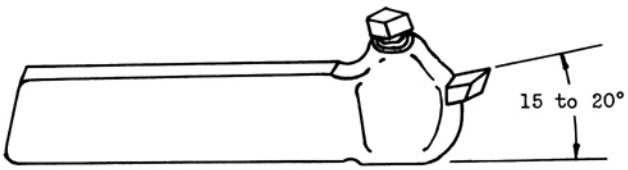


Fig. 473

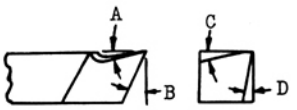


Fig. 472

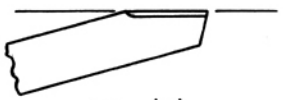


Fig. 474

Fig. 475

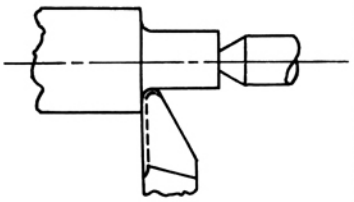


Fig. 476

All cutting tools should be ground with a wet grinding wheel, or if ground dry, they should be frequently dipped in water to prevent annealing of the cutting edge.

Four angles are important in grinding tools. These angles, shown in Fig. 472, are the top rake (A), side rake (C), front clearance (B), and side clearance (D). The angles vary with the material being machined. (NOTE. The rake angles deal with the top of the tool bit and the clearance angles deal with the side and front.)

The top rake is usually provided for in the tool holder by setting the tool on an angle. This angle is usually 15° but it may be as high as 20°, as shown in Fig. 473. These rake angles are correct for the machining of steel and cast iron, but on forged tools it is necessary to grind the top rake in the tool. This top rake can be varied some to suit the material being turned by adjusting the tool in the post through the rocker. The softer the material the less the top rake should be, as there is a tendency for the tool to dig in if the rake is too great.

There should be no top rake for turning bronze, as shown in Fig. 474, and the cutting edge of the tool should be about horizontal. A negative rake (Fig. 475) is often used for turning soft copper, babbitt, and some die casting alloys.

The side rake (Fig. 472, C) also varies with the material being machined. With this angle large enough, the tool will drag the carriage along by feeding into the work of its own accord, especially if the material is soft. The tool would not cut without the side rake and the feed mechanism would be under excessive strain. The proper angle is from 6° for soft material to 15° for steel. A tool is ground with side clearance (Fig. 472, D) to take care of the feed advance and prevent the tool from dragging on the shoulder formed by the cut (see Fig. 476). This angle is about 6° from the vertical and is constant.

The front clearance (Fig. 472, B) depends somewhat on the diameter of the work to be turned. If a tool were ground square, without any front clearance, it would not cut, as the material being turned would rub on the cutting tool just below the cutting edge. The front clearance is necessary to eliminate this interference. This clearance, which should be less for small diameters than for large diameters, may range from 8° to 15°. Do not grind more front clearance than is necessary, as this takes away the support from the cutting edge of the tool.

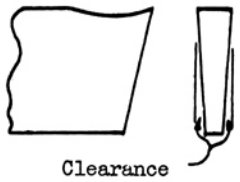


Fig. 477

Tool bits should not be ground in their holders.

After a tool has been ground on a grinding wheel, it will produce better work and last longer if the cutting edge is stoned with an oil stone. This takes away the wheel marks and gives a smooth cutting edge. In grinding cutting off tools, be careful to see that both sides of a tool have the necessary clearance. A tool of this kind cuts better if the lip is ground back of the cutting edge (see Fig. 477), to curl the chip as it comes off the work.

The cutting edge of a tool must be set on the center line of the work, as shown in Fig. 478. This illustration also shows a tool properly ground and set for cutting the harder metals, such as tool steel and high carbon steel. Note the top and side rake.

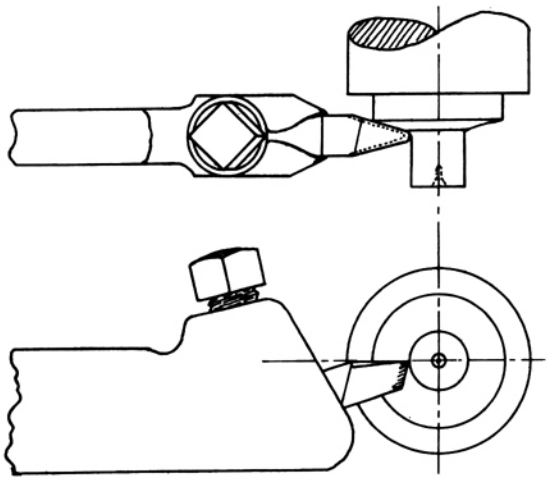


Fig. 478

The helix angle is the angle made by the helix of the thread at the pitch diameter with a plane perpendicular to the axis. Because of the helix angle, the square threading tool cannot be ground like the cutting off tool. The part of the tool bit which supports the cutting edge must be ground to the helix angle so that the bottom of the tool will not rub against the sides of the thread. The helix angle is found by the method shown in Fig. 480. The line FG equals the lead of the thread. EF equals the circumference of the pitch diameter and is drawn at right angles to FG. Angle B is the helix angle to which the tool must be ground. The tool must also be given clearance on the sides, as shown in Fig. 480.

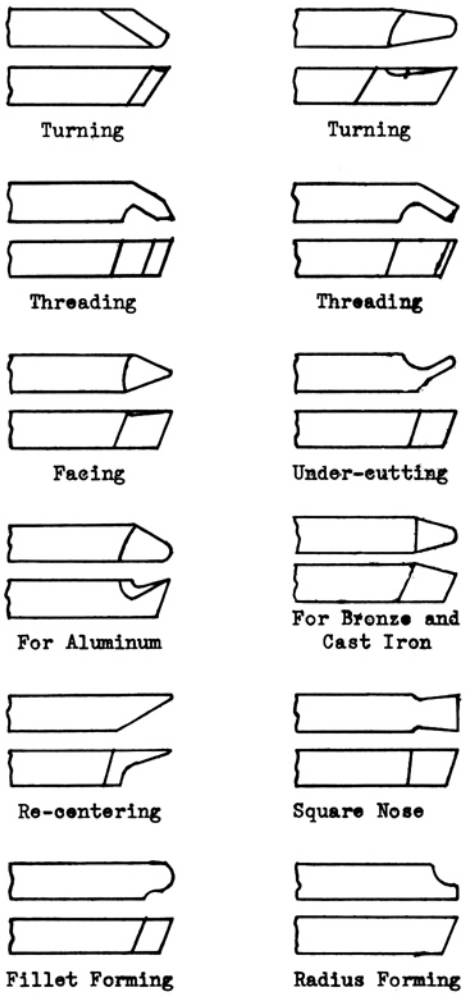


Fig. 479. Shapes of Turning Tools

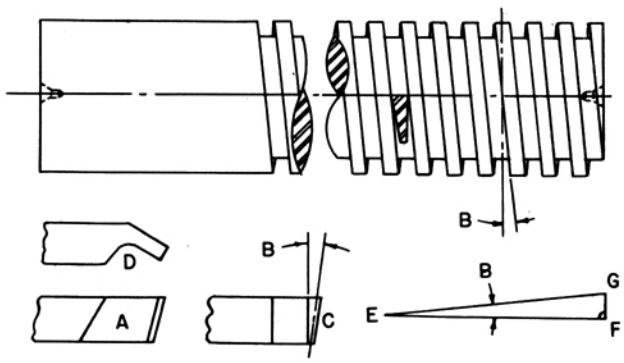


Fig. 480

In grinding boring tools see that the front clearance is sufficient to prevent the tool from rubbing in the hole and dragging at the point shown in Fig. 481. The boring tool should be ground as shown in Fig. 482. The front clearance angle of shaper and planer

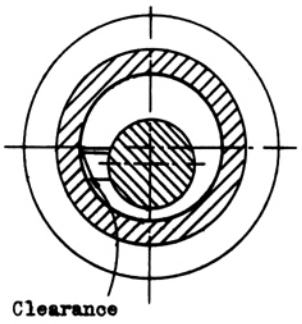


Fig. 481

cutting tools should be 3° to 4° when they are in the cutting position (see Fig. 483).

The tool bits shown in Fig. 484 can be used to shape and plane most jobs. If a job is an unusual one, it may be necessary to grind the form a little different from the ones shown in this illustration.

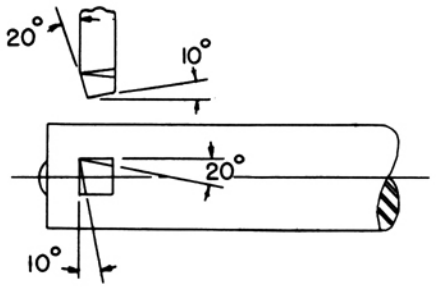


Fig. 482

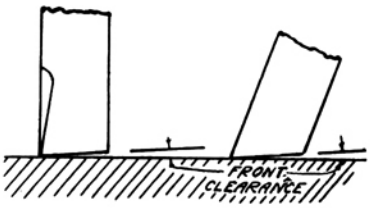


Fig. 483

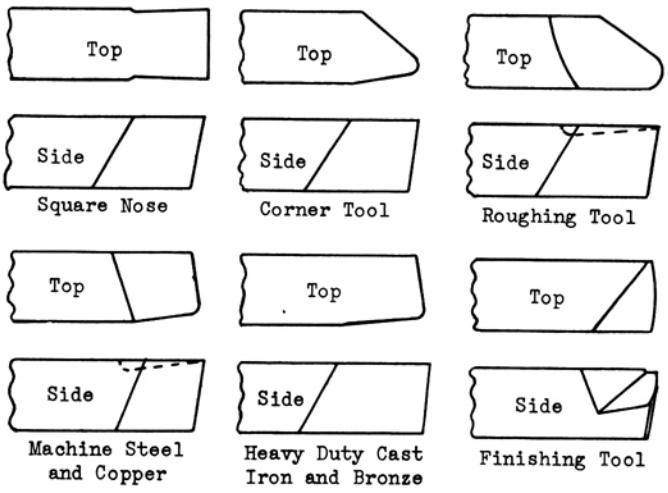


Fig. 484. Common Types of Shaper and Planer Bits

Chapter 17

SHAPER

1. What is a shaper?

A. A shaper is a machine that has a ram with a reciprocating motion. The ram holds the apron and tool. A chip is peeled off the work on the forward stroke. An adjustable table with T slots holds the work, vise, and other fixtures for holding the work. Fig. 485 shows one of the types of shapers used in the Trade School shop.

Shapers are divided into two distinct

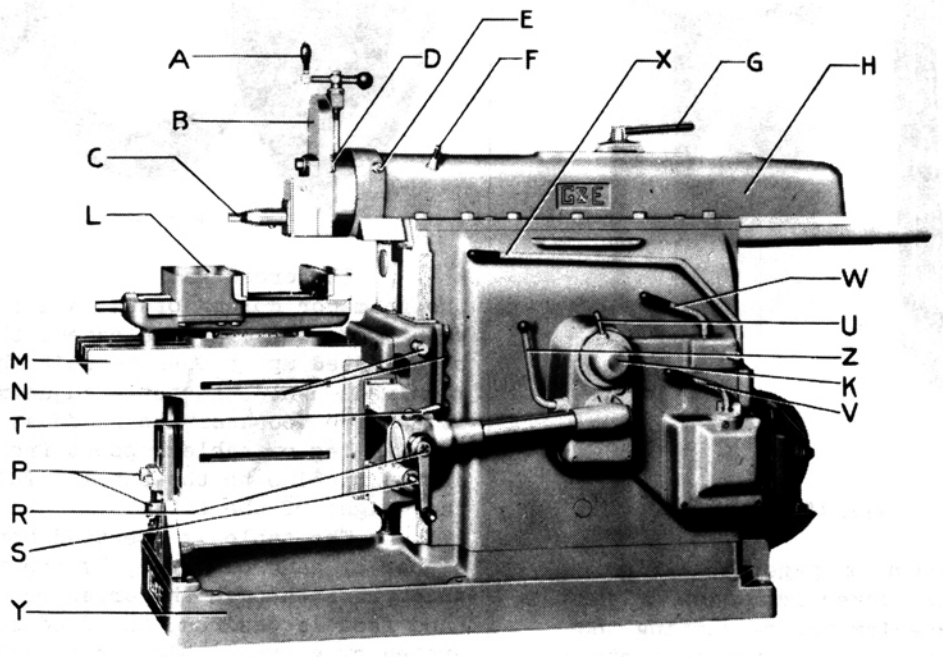


Fig. 485
Nomenclature of Gould and Eberhardt Shaper

A - TOOL HEAD SLIDE CONTROL - For positioning slide vertically.
B - TOOL HEAD SLIDE
C - TOOL POST
D - TOOL SLIDE LOCK - For clamping slide in vertically adjusted position.
E - TOOL HEAD LOCK - For clamping head in adjusted position.
F - RAM POSITIONING CONTROL
G - RAM CLAMP - Clamp ram to link and lever after positioning.
H - RAM
K - STROKE LENGTH CONTROL - Automatically locked. Positions crank pin in crank plate to obtain stroke lengths.
L - VISE
M - REGULAR BOX WORK TABLE - Called a Universal Table when constructed to swivel 360° and to have one surface that tilts.
N - RAIL CLAMP - For locking cross rail in adjusted position.
P - TABLE SUPPORT CLAMP AND LOCK
R - TABLE HORIZONTAL POSITION CONTROL - For positioning and feeding table manually.
S - TABLE VERTICAL POSITION CONTROL - For raising and lowering work table.
T - FEED DIRECTION CONTROL - Neutral position disengages feed.
U - FEED SELECTOR
V - TRANSMISSION SPEED SELECTOR - For obtaining eight ram speeds.
W - BACK GEAR SELECTOR - Used in conjunction with V.
X - CLUTCH CONTROL - For starting and stopping. Operates brake when stopping machine.
Y - BASE
Z - POWER RAPID TRAVERSE CONTROL - Moves table horizontally in direction opposite to feed set.

classes, namely, crank and geared. The most common type found in the toolroom is the

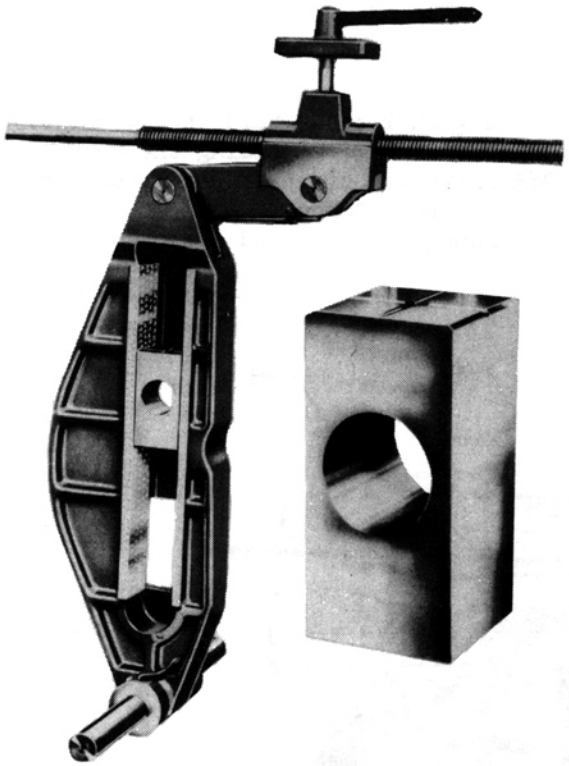


Fig. 486

crank shaper, shown on page 115. Figs. 486 and 487 show the rocker arm which drives the ram, and the mechanism for regulating the length of the stroke. The ram supports the tool head. The head carries the downfeed mechanism and will swivel from side to side to permit the cutting of angles. This is generally a hand feed, but some shapers are equipped with a power downfeed in addition to the regular hand feed.

The table of the shaper is of box form with "T" slots on the top and sides, for clamping work. Some shapers have a universal table, shown in Fig. 489, which can be swiveled through an arc of about 180°. This type of table also has a top which can be tilted approximately 15°.

The apron is fastened to the front of the shaper head and consists of the clapper box, tool block, hinge pin, and tool post. It can be made to swivel, and when taking vertical or angular cuts the upper part of the apron is turned in a direction away from the surface of the cut. In this position the tool will not dig into the work, but will have a tendency to hinge in a direction away from the cut.

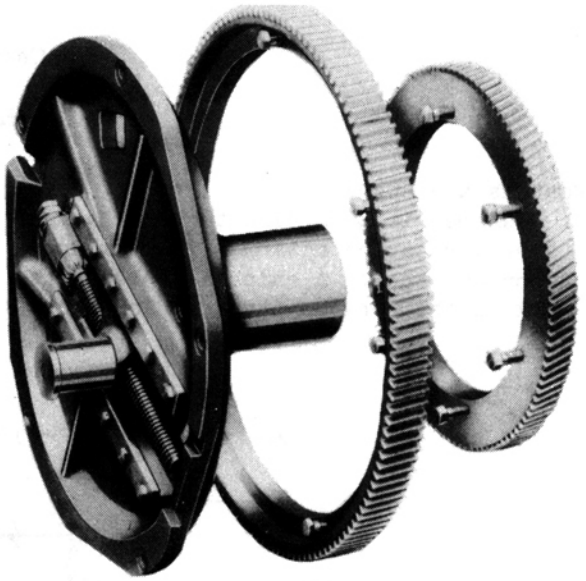


Fig. 487

The cross rail is bolted directly to the frame or column of the shaper with bolts. When these bolts are released, the table may be adjusted up or down to suit the job being machined. When the table adjustment is made it is also necessary to readjust the supporting leg or table support from the base of the machine to the table. This supports the weight of the table.

The table is operated horizontally, either by hand or power, by the feed screw which runs through the cross rail. When the hand feed is desired, the feed screw is operated by a crank which is placed on the squared end of the screw. The automatic feed or power feed is obtained by a pawl which engages in a notched wheel or ratchet. This is either fastened to the feed screw or transmits its motion to the screw through gears. This mechanism causes the feed screw to make part of a revolution each time the ram makes a stroke. On the crank shaper the feed screw should always be made to operate on the return stroke of the ram.

2. How is the size of the shaper determined?
   A. By the size cube that it will shape. A 24-inch shaper, for example, has a stroke and transversal table feed of 24 inches. The downfeed, however, is only a few inches and the cube must therefore be turned each time a new surface is shaped.

3. What is the difference between a Plain and a Universal shaper?

A. The main difference is the type of table each has. Fig. 488 shows the table of the plain shaper. This table cannot be swiveled or tilted. Fig. 489 shows the table

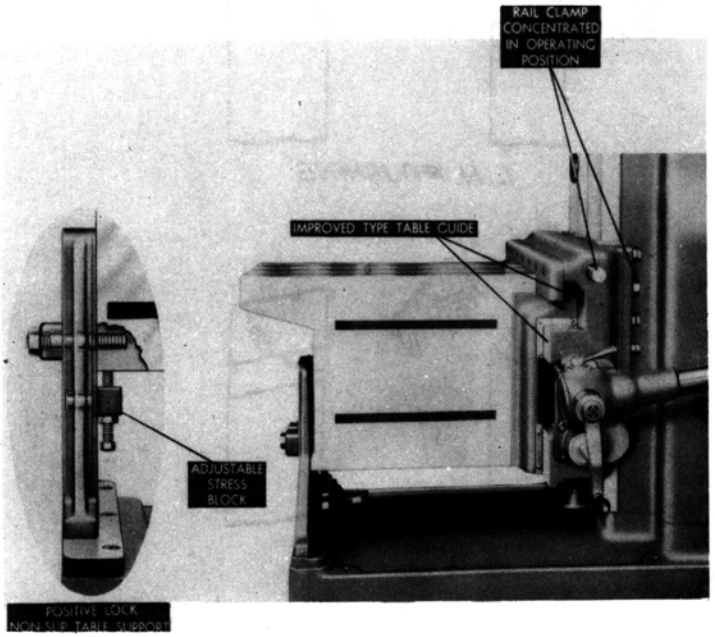

RAIL CLAMP CONCENTRATED IN OPERATING POSITION
IMPROVED TYPE TABLE GUIDE
ADJUSTABLE STRESS BLOCK
POSITIVE LOCK NON-SLIP TABLE SUPPORT

Fig. 488

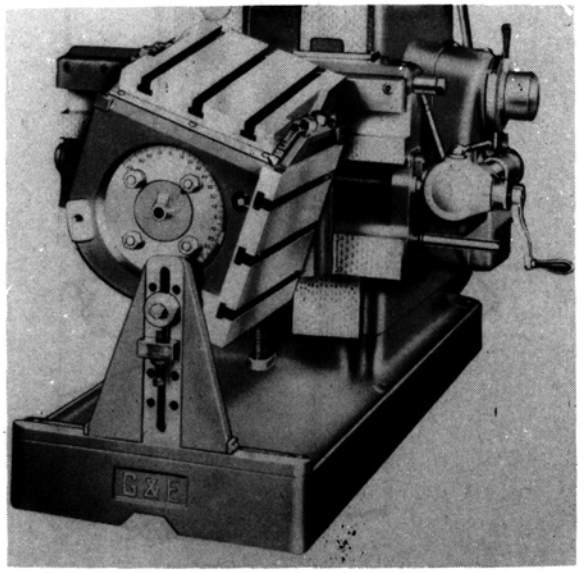


Fig. 489

of a universal shaper. This table can be tilted to 15° and swiveled through an arc of 180°. The universal type is commonly used in toolrooms for machining angular cuts.

4. What are some of the first things to do before starting a cut on a shaper?
   A. Be sure that you thoroughly understand the blue print and work; see that all tools are clear of working or moving parts, that the job is held securely, that the machine is well oiled and in condition to operate, that parallels are both of the same height, and that you have enough stock.

5. What are three essential things to remember while operating a shaper?
   A. Wear your goggles, always be careful, and keep your mind on your work.

6. What is the first step in machining a job having several cuts and angles?
   A. Check the sketch or blue print and see if the job is understood clearly.

7. What is the purpose of layout lines on a job?
   A. Layout lines are generally used as a guide to show the amount of material left for machining. The job must be machined to the dimensions given on the blue print. If the metal is a casting on which the line has been scribed from the outline of a template, then in machining the tool point must split the layout line. On a job of this type, consult your instructor or foreman.

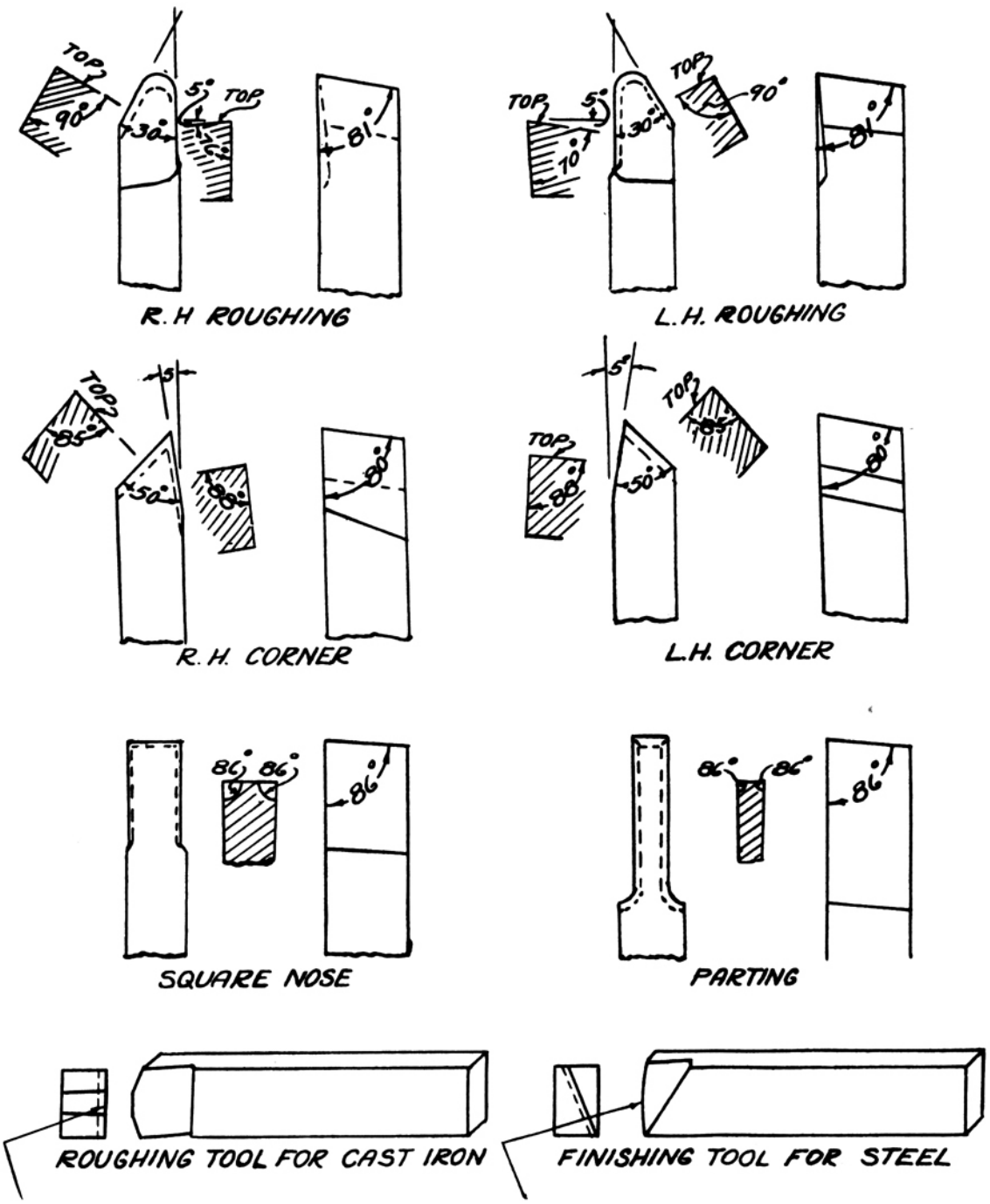
8. Sketch some shaper and planer tool bits.
   A. See Fig. 490.

9. In what respect does a shaper or planer tool differ from a lathe tool?
   A. Owing to the fact that the shaper tool feeds into the work on the return stroke only, the side clearance angle need not be nearly as great as that of the lathe tool, which continually feeds into the work.

10. How should the tool holder be set in the tool post?
    A. Set the tool holder so that the tool bit does not extend farther than two inches from the tool post, to avoid chattering.

11. How much clearance should there be between the work in the vise and the ram?
    A. About two inches, or enough to clear the hand.

12. What tools are used to hold work squarely in the vise?
    A. Use parallels or hold-downs. Place true side of work against the solid jaw, and insert a parallel or hold-down between the



Tool with broad cutting edge for cutting cast iron, using coarse feed. Wide cutting edge gives a smooth finish.

Tool with slightly rounded cutting edge for cutting steel, using coarse feed. Rounded cutting edge gives a shearing cut.

Fig. 490

other side of work and movable jaw.

13. State the operations in shaping a rectangular job.

A. Shape the thickness first, then the width, and then the length. After shaping the thickness, it is necessary to shape the width square with the finished surface. Place job in horizontal plane, one side of the thickness against the solid jaw. Then place parallel or hold-down against the other side and tighten vise. After the width has been

squared with the thickness, place the finished width on parallels and hold with holddown, forcing job against solid jaw.

Next, the job must be squared and shaped to length. Swivel vise to 90° with stroke of ram and place job in end of vise. The end to be shaped should be placed in the horizontal plane. The other end of the vise must be blocked with a piece of material of like thickness, to maintain parallelism of vise jaws and to hold the work more securely. Place width of job parallel with stroke of ram; the solid jaw will hold job square in this direction. With the aid of the solid square placed on the bottom of the vise and against the work, true work and clamp tightly. This will bring end of job square with thickness.

14. How should the tool be started when taking the first cut?

A. The shaper is started and the length of the stroke adjusted for length and position, to give the tool a movement somewhat greater than total length of work. The tool is fed downward into the work and the latter is moved crosswise by hand until the desired depth is started. The automatic feed is then engaged by dropping the feed-pawl into mesh with the ratchet gear on lead screw.

15. On what stroke should the table feed?

A. The table should feed on the return stroke.

16. What are the most common causes of chattering?

A. Tool clearance too great, tool suspended too far from tool holder, work not held rigidly in vise, and ram gibs not in proper adjustment.

17. How can chattering be eliminated?

A. By either of the following adjustments: Sharpen the tool with from 2° to 3° clearance and clamp the work rigidly in the vise, or see that the gibs are properly adjusted.

18. Explain how to set the stroke of ram for length and for position.

A. When setting length of stroke, bring ram to the extreme back position and set for length of work plus about 5/8 of an inch as shown on the graduated scale on machine. This will allow for about 1/8 of an inch clearance in the front of the work and 1/2 of an inch in the back. For position of stroke, bring ram to extreme forward position and adjust it so that front of tool bit will clear front of job by about 1/8 of an inch.

19. How may the solid jaw of the vise be checked for being square and parallel

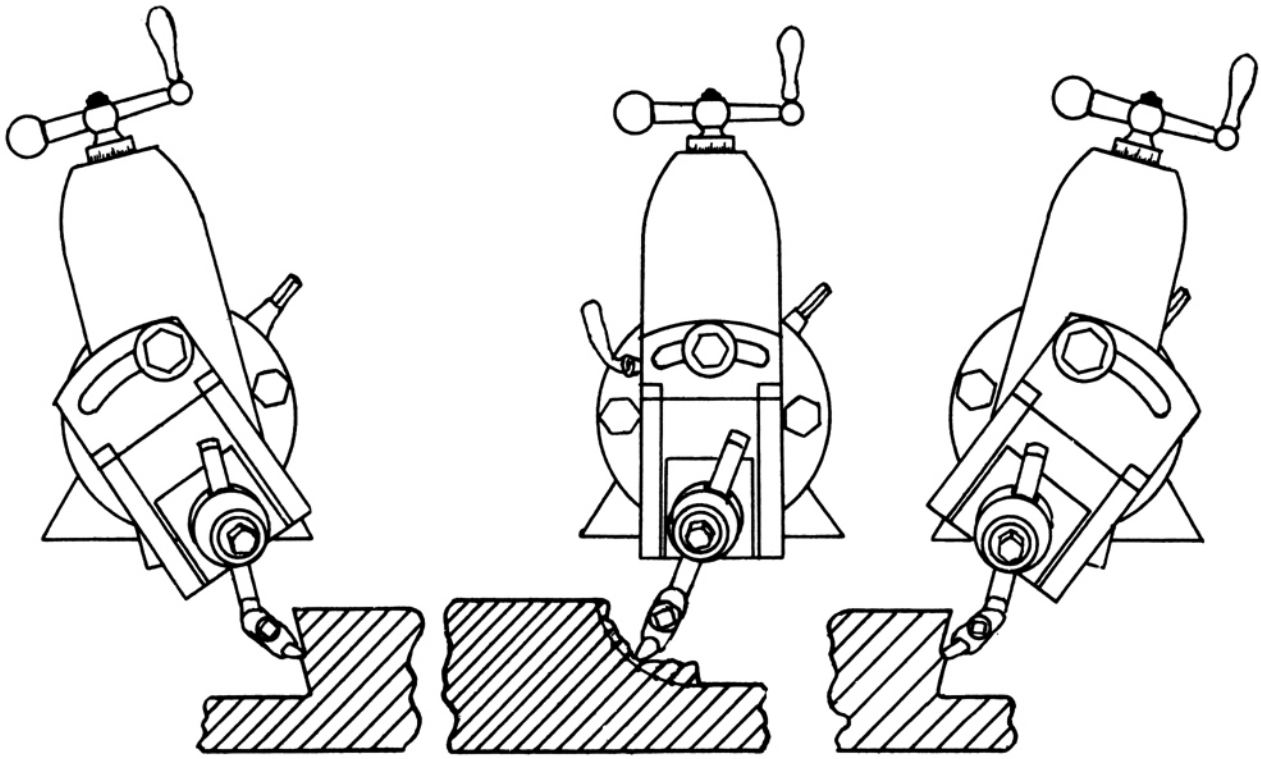


Fig. 491

with stroke of ram?

A. To check solid jaw for being square with stroke of ram, place an indicator in tool holder with point of indicator on solid jaw, and move the vise at right angles to the ram with cross feed. To check solid jaw for being parallel with stroke of the ram, place an indicator in the tool holder with point of the indicator against solid jaw and move ram forward and backward.

20. What should be done if work is not parallel within reasonable limits after a cut has been taken from each side?

A. Inspect the vise thoroughly for dirt and chips and make sure that it is clean. If this does not correct the condition check the vise with an indicator.

21. Draw sketches showing how to set head and clapper box when taking vertical and angular cuts.

A. When making angular or vertical cuts on a shaper the clapper box must be set at an angle so that the tool will swing away from the work on the return stroke of the ram. See Fig. 491.

22. Explain how a dovetail bearing may be cut on a shaper.

A. The toolhead of the shaper should be set over at an angle the same as the angle of the dovetail to be cut. When dovetail bearings such as those shown in Fig. 492 are to be cut the work should not be disturbed in shaping the angular and flat surfaces of the dovetail Aa, Bb, and Cc. Surfaces Bb and Cc should be machined before completing the angular surfaces Aa. A right-hand tool and a left-hand tool are used to machine the angular sides, as shown in the illustration. A roughing and a finishing tool should be used if considerable stock is to be removed. In using two tools and moving the tool head from one side of the center line to the other if there is any variation in the angular setting of the head, a variation in the angular sides of the dovetail will result.

Another way of cutting a dovetail, when the sides of the work are parallel and the solid jaw of the vise is parallel with the stroke of the ram, requires only one tool. First, rough out the sides of the dovetail to within 1/64 or 1/32 of the finished size. Next, take a light cut on one side, reverse the work in the vise but do not disturb the setting of the table, and take a light cut off the other side. Then check for size and repeat the process until the finished size is reached. Using this method the dovetail will be held central with the work and the angles will be the same.

In shaping dovetail bearings it is very important to incline the clapper box in the proper direction so that the tool will swing away from the work on the return stroke of the ram. The beginner should pay strict attention to this point because sometimes the setting may not be correct although it may appear to be. Remember that the top of the clapper box must be set in a direction away from the surface being machined (see question 21).

23. How should a dovetail be checked for size?

A. A dovetail slide bearing, as the name implies, derives its name from the shape of a dove's tail. It is used a great deal in the modern toolroom on parts of machines such as the ram and cross slide on the shaper, compound slide, and taper attachment on the lathe, etc. Dovetail bearings are usually machined on a shaper or planer but may also be made on a milling machine. When a gib (a piece located alongside a sliding member to take up the wear) is used, the accuracy required in planning is not as great as when the two pieces are fitted together. In either case, however, a smooth cut is necessary and one or two thousandths of an inch should be left for scraping. It is good practice to lay out the dovetail before starting the work.

In order to check the size of a dovetail to determine the amount necessary to machine off or to check the finished product

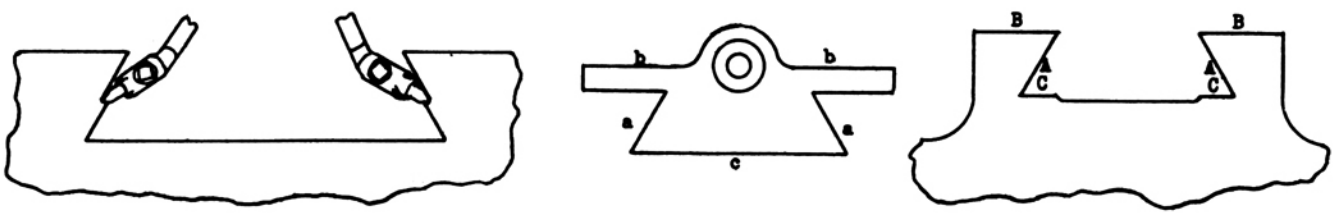


Fig. 492

accurately, two plug gages are placed in the angles of the dovetail and a measurement is taken either over or between the plugs. This measurement may be determined by applying the rule from trigonometry which states that the length of the side adjacent in a right triangle is equal to the length of the side opposite multiplied by the cotangent of the given angle. The side adjacent is that side which with the hypotenuse makes the acute angle in question. The side opposite is the side which is opposite the angle in question. The cotangent is the ratio existing between the side opposite and the side adjacent. The example below shows the method used to find the required measurement.

24. When cutting an angle on a large job, using a shaper which has a universal table, should the table be tilted, or the tool head swung to the required angle?

A. The universal table should be tilted because this makes it possible to use the automatic table feed. If the tool head were set on the required angle and did not have an automatic down feed, it would be necessary to use the hand down feed.

25. Explain the method used to shape a "V" or keyway central in a block.

A. One method used is to lay out the job and shape to scribed layout lines. A more accurate method of shaping a "V" block is to

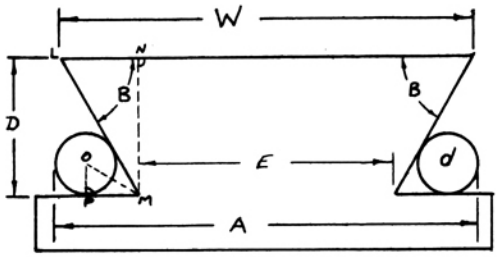


Fig. 493. Internal Dovetail

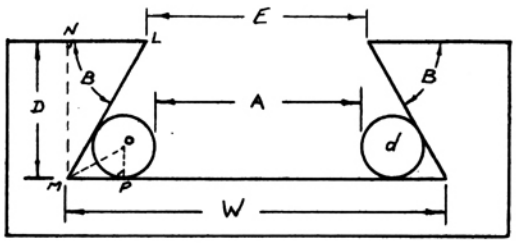


Fig. 494. External Dovetail

To solve for A (Fig. 493).

$$E = W - 2(LN)$$
$$A = E + 2MP + d$$

Example

$W = 4.5$
$d = .250$
$D = .750$
$B = 60^\circ$

Solution

$LN = .75 \times \cot 60^\circ = .43301$
$E = 4.5 - (2 \times .43301) =$
$4.5 - .86602 = 3.6339$
$OP = .250 \div 2 = .125$
$MP = .125 \times \cot 30^\circ =$
$.125 \times 1.732 = .2165$
$A = 3.6339 + (2 \times .2165) + .250 =$
$3.6339 + .433 + .250 = 4.3169$

To solve for A when E is given (Fig. 494).

$$W = E + 2(LN)$$
$$A = W - (2MP + d)$$

Example

$E = 3.634$
$d = .250$
$D = .750$
$B = 60^\circ$

Solution

$LN = .75 \times \cot 60^\circ = .43301$
$W = 3.634 + (2 \times .43301) =$
$3.634 + .86602 = 4.5$
$OP = .250 \div 2 = .125$
$MP = .125 \times \cot 30^\circ =$
$.125 \times 1.732 = .2165$
$A = 4.5 - (2 \times .2165) - .250 =$
$4.5 - .683 = 3.817$

Distance measured over plugs = A + 2d

The following formulas and table of values are given for the most commonly used forms of dovetails.

$$MP = OP \times \text{Cot } \frac{B}{2}$$

$$E = W - 2(D \times \text{Cot } B)$$

$$A = E + 2MP + d$$

Angle B	15°	$17\frac{1}{2}^\circ$	20°	$22\frac{1}{2}^\circ$	25°	$27\frac{1}{2}^\circ$	30°	35°	40°	45°	50°	55°	60°
Cotangent	3.732	3.1716	2.7475	2.4142	2.1445	1.9210	1.732	1.4281	1.1917	1.0000	.83910	.70021	.57735

set the vise jaws parallel to the stroke, set the tool head at the required angle, and rough out to the layout lines. Next, take a cut from one side of the "V", then reverse the job in the vise. With the table set in the same position, take a cut off the opposite side. Continue this until both sides of the "V" are cleaned up and the proper depth has been reached. The "V" will then be in the center of the block.

26. Tell how to machine a job similar to the one shown in the sketch at the right.

A. First machine the six sides following the instructions given in question 13. Next, lay out the angle and the step according to the required dimensions. See Fig. 495. Place the work in the vise on parallels, swivel the vise to 15°, and rough out. Complete the machining by setting the tool head at 7°, feeding the tool down to the layout on this angle and finishing the thickness of the step in one operation.

27. How may a keyway be cut on a shaper when the keyway does not extend the entire length of the shaft?

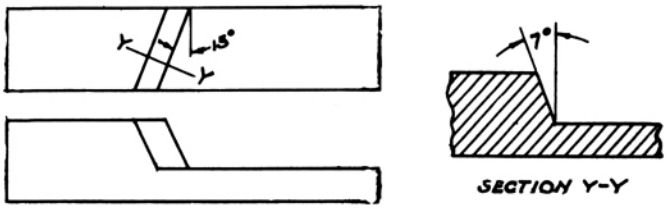


Fig. 495

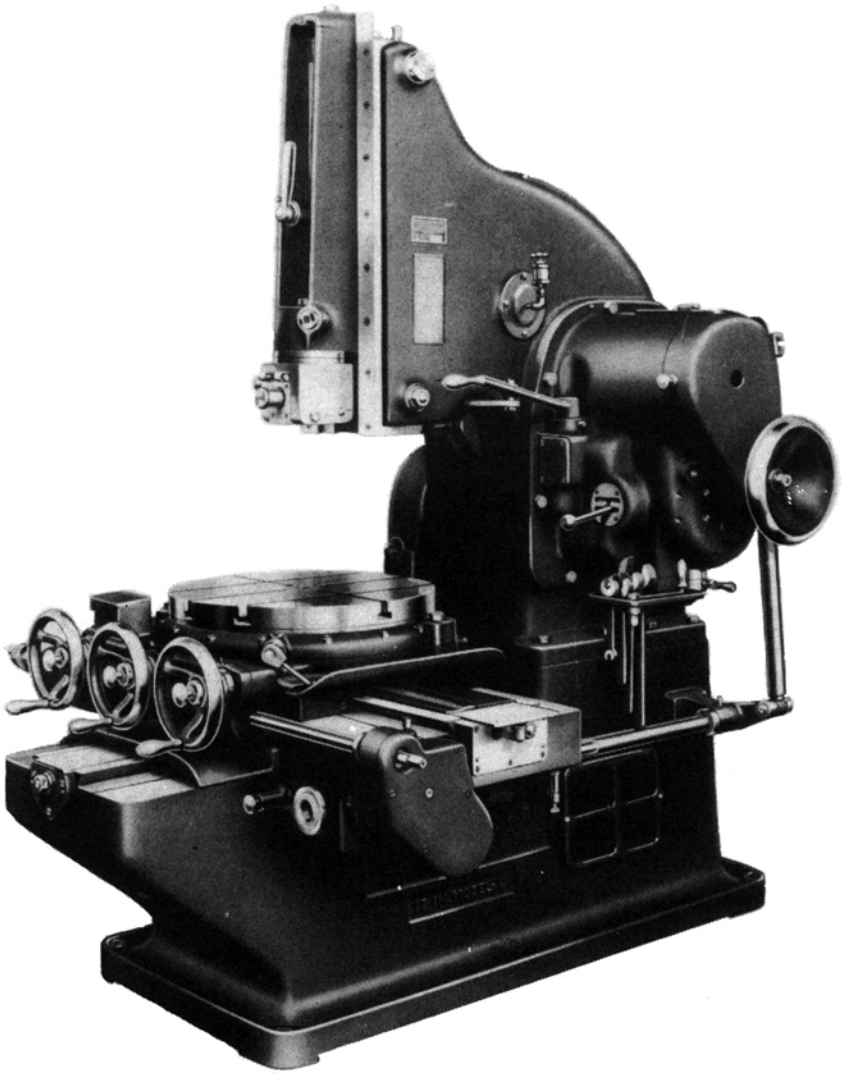


Fig. 496
Vertical Shaper

A. Drill a hole slightly wider and deeper than the width and depth of the keyway at the place where the keyway ends. Set the position of the shaper stroke so that the tool will stop in center of the drilled hole at the end of the forward stroke. Then cut the keyway to size in the usual manner.

28. How should a thin job be machined to prevent warping?

A. First a light cut should be taken off each side to relieve the internal strain. Then light cuts should be taken off each side alternately until the correct thickness is reached.

29. Why should the speed of the shaper be increased when taking a finishing cut?

A. The speed should be increased because this gives a better finish and shortens the length of time for machining.

30. What tool is the most important for checking the squareness of shaper work?

A. The solid square is the most important.

31. Why is it unnecessary to hammer the shaper vise handle when clamping work?

A. The vise handle is long enough so that the leverage gained when pressure is exerted on the end of it is sufficient to clamp the jaws tightly on the work.

32. Describe the slotter.

A. Vertical slotters were originally designed for simple operations of slotting or key seating. They are still simple rugged machines primarily adapted for such simple operations.

33. Describe the vertical shaper.

A. The vertical shaper (shown in Fig. 496) is more commonly used than the slotter. It is a machine having all the design features of the vertical slotter and in addition has a swiveling tool holder and an angularly adjustable ram. By using this adjustable ram cuts can be taken at an angle with the vertical. This feature is used mostly for shaping clearance on dies, etc.

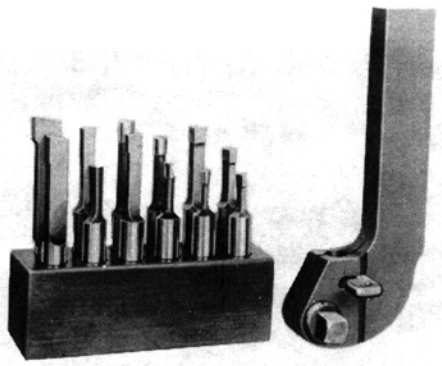


Fig. 497. Tools and Holder

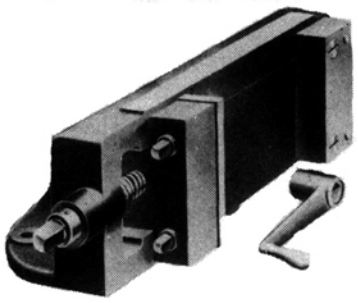


Fig. 498
Vise

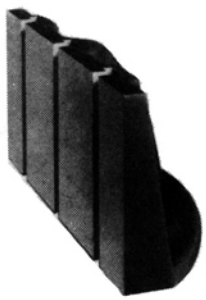


Fig. 499
Angle Iron

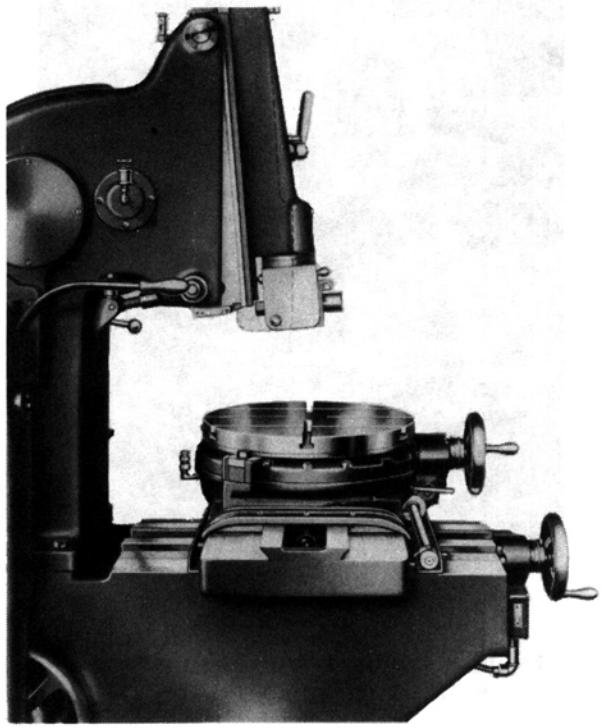


Fig. 500
Showing Angular Adjustment of Ram

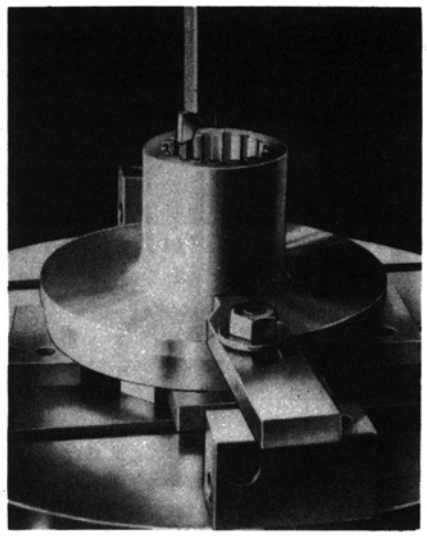


Fig. 501. Internal Splining

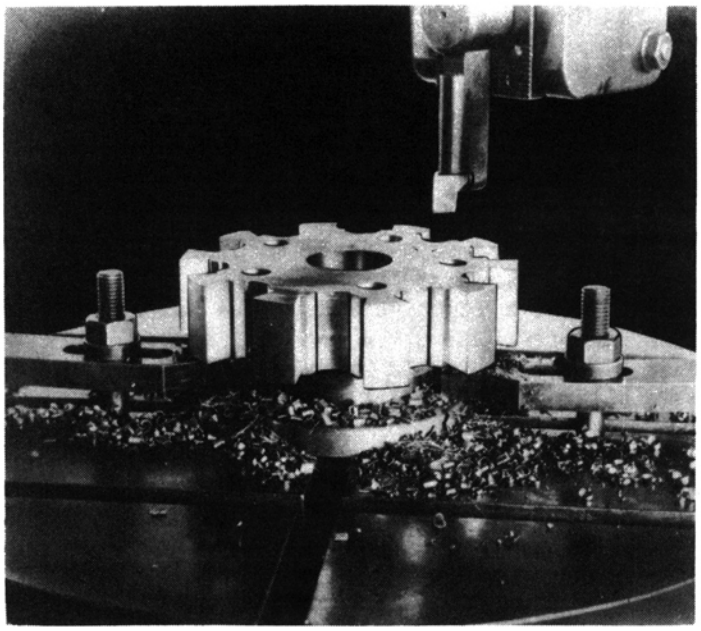


Fig. 502
Machining a punch for producing laminations.

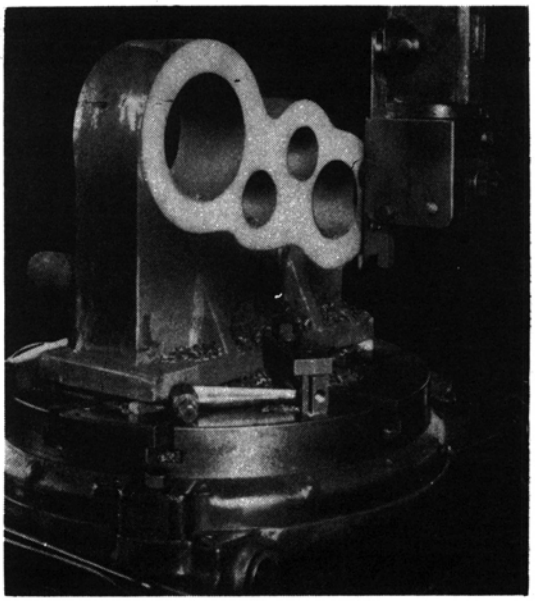


Fig. 503
Facing a casting on both sides without a change in set up.

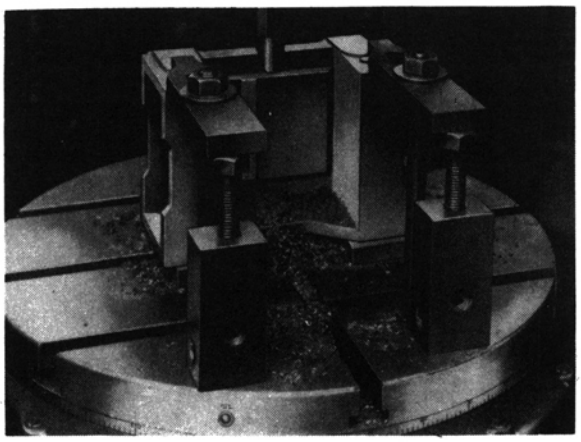


Fig. 504. Machining A Box Jig.

Chapter 18

PLANER

34. What is the function of the planer?

A. The planer (shown in Fig. 505) is for producing flat surfaces on work that is too large or otherwise impracticable or impossible to machine on the milling machine or shaper.

35. What is the main difference between a shaper and a planer?

A. With the planer the table or platen has a reciprocating motion past the tool head, while on the shaper the table is stationary and the tool head, which is fastened to the ram has the reciprocating motion.

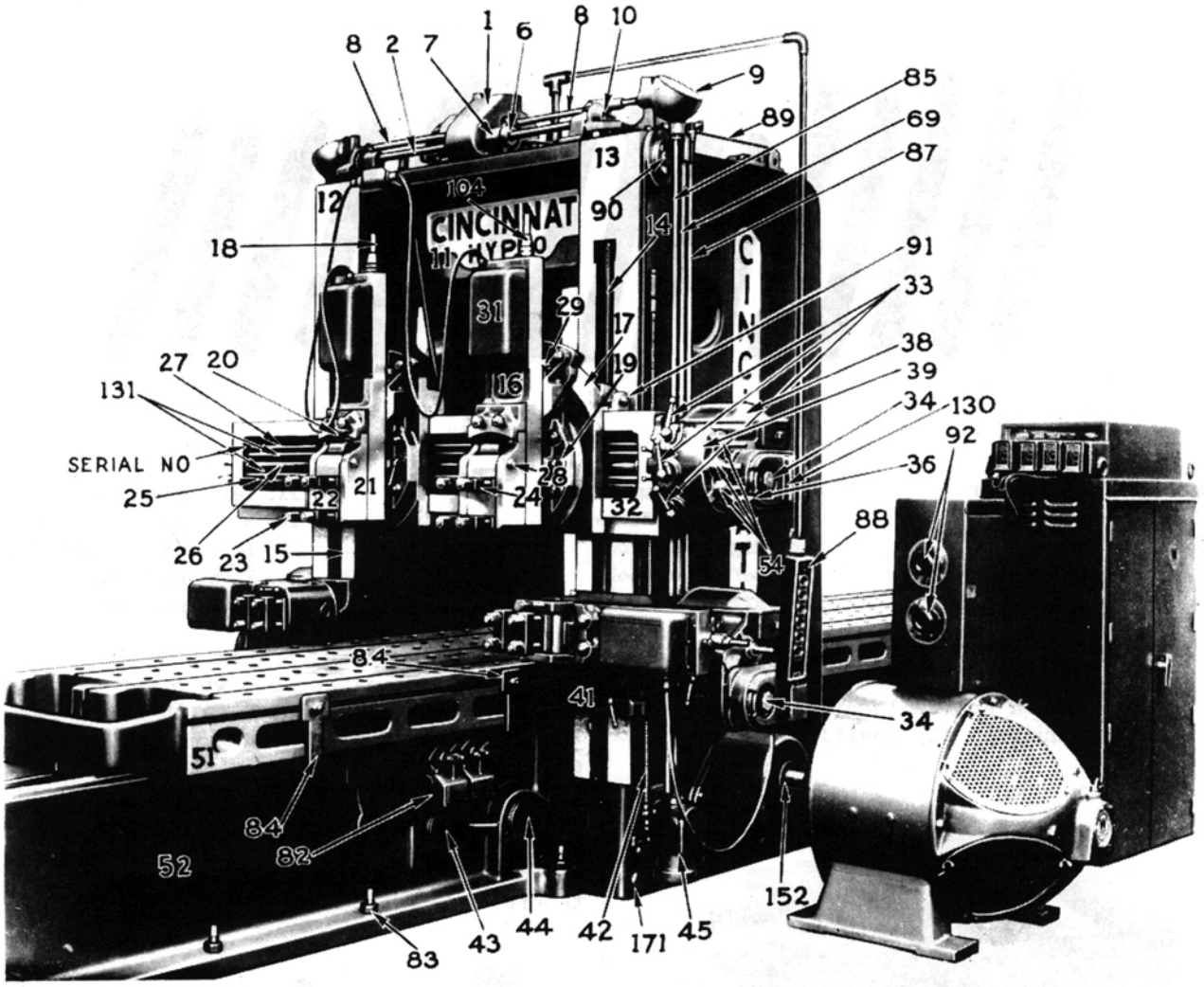


Fig. 505. Planer

1. Elevating Device
2. Horitonzal Elevating Shaft
6. Elevating Shifter Fork
7. Elevating Clutch
8. Horizontal Rapid Traverse Shaft
9. Rapid Traverse Bracket
10. Elevating Bracket
11. Arch
12. Left-Hand Housing
13. Right-Hand Housing
14. Right-Hand Elevating Screw
15. Left-Hand Elevating Screw
16. Slide
17. Saddle
18. Down Feed Screw
19. Harp
20. Clapper Box Clamp
21. Clapper Box
22. Tool Block
23. Tool Block Stud
24. Tool Block Clamp
25. Rail Screw, Right-Hand Head
26. Feed Rod
27. Rail Screw, Left-Hand Head
28. Clapper Box Taper Pin
29. Harp Gib
31. Right-Hand Head complete
32. Rail
33. Rapid Traverse and Feed Engaging Lever
34. Feed Changing Knob
36. Worm Shaft or Rail Clamping Shaft
38. Rail Clamp Feed Cover
39. Rail Clamp Cover
41. Side Head complete
42. Side Head Screw
43. Bull Wheel Shaft
44. Bull Pinion Shaft
45. Feed Drive Box
51. Table
52. Bed
54. Feed Reverse Knob
90. Feed Shaft
82. Table Reversing Master Switch
83. Leveling Screw
84. Table Dogs
85. Rapid Traverse Shaft
87. Rod, Interlocking Device
88. Pendant Station
89. Counterweight Cable
90. Counterweight Sheave Wheel
91. Top Saddle Gib
92. Cut and Return Speed Rheostats
104. Down Feed Screw Micrometer Collar
130. Cover Measuring Unit
131. Shifter Rod
152. Coupling Main Motor
171. Housing Bolt

36. How is the size of the planer designated?

A. The size of a planer is designated by the distance between the uprights, the distance between the platen and the cross rail at its highest position, and the maximum stroke. For example, 24" x 24" x 6'.

37. Describe a method of ascertaining if the cross rail is parallel to the platen.

A. Clamp an indicator on the tool box, with the point touching the platen, and note the indicator reading as the head is moved the length of the cross rail.

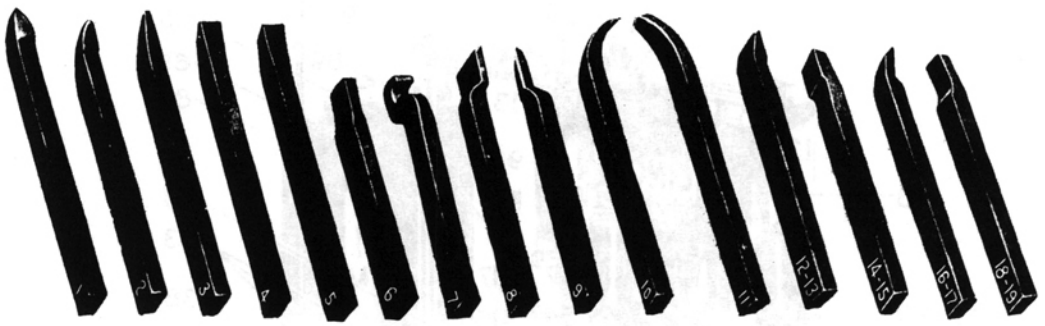


Fig. 506. Planer Tools

1. Right-Hand Rougher
2. Left-Hand Rougher
3. Round Nose
4. Square Nose
5. Square Nose
6. 3/8 in. Square Nose
7. Gooseneck Finisher
8. Right Side Tool
9. Left Side Tool
10. Right Side Rougher
11. Left Side Rougher
12. Right Angle or Side Tool
14. Left Angle or Side Tool
16. Right Angle Tool
18. Left Angle Tool

38. The planer chuck shown in Fig. 507 is mounted upon a swivel base which is provided with a split clamping ring for securing chuck in any position. A tapered pin is provided for locating the chuck either parallel or at right angles with the base. The base has a slot at each side for securing it to the platen of the planer.

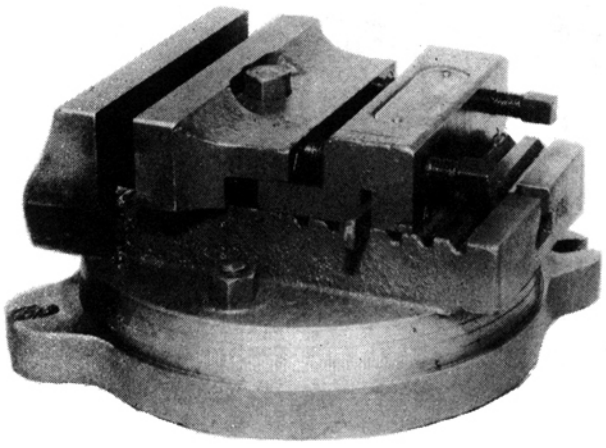


Fig. 507. Planer Chuck

39. Describe the planer centers.

A. Planer centers are provided with an index consisting of circular plates or disks which have notches accurately spaced on their periphery and give a wide range of divisions. See Fig. 508.

The work, together with the index, is revolved by means of a worm and gear, which can readily be disengaged when not required.

Tongues are inserted in the bottom of the headstock and tailstock, which can be taken out and fitted to the slot in any planer by planing or filing off one side, without destroying the alignment. The footstock is inserted in a block which is adjustable up and down for convenience in planing bevels or tapers.

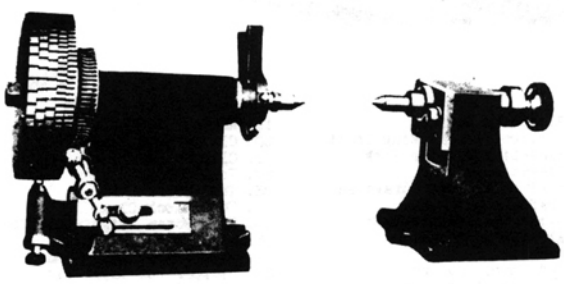


Fig. 508
(Left) Headstock; (Right) Footstock

40. Give some hints on clamping work to the planer table.

A. In clamping work on the planer table the operator must take great care to see that the work is fastened securely by clamps, bolts, poppets, toe-dogs, etc., (see Figs. 509 to 516).

Clamps should not be placed on a finished part of the work unless it is protected with copper, brass, paper, etc.

Flat thin work is held down best by toe-dogs and poppets.

See that the work does not spring when tightening clamps or toe-dogs.

A washer must be used with every bolt and nut. Select bolts of the proper length.

An open end wrench of the correct size should be used on all square and hexagon nuts.

When placing the work in position see that it does not mar the finished surface of the table. Allow enough clearance for the work to pass under the cross rail and by the uprights.

Place thin paper under the work to seat it properly.

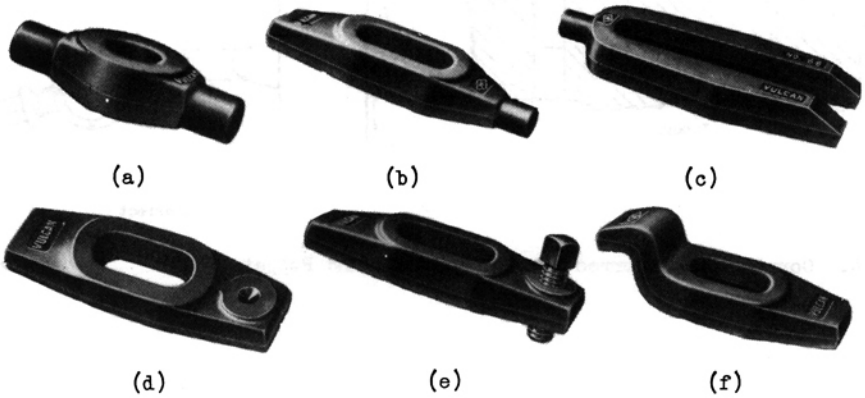


Fig. 509. Clamps: (a) Finger, Double End; (b) Finger, Single End; (c) "U"-Pattern; (d) Plain Slot; (e) Adjustable Step; (f) Gooseneck.

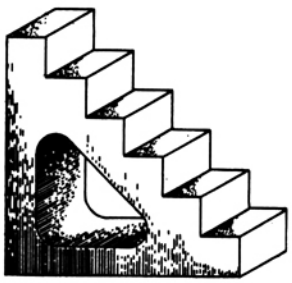


Fig. 510. Step Block

Fig. 511. Planer Jack

Fig. 512. Setting Planer Gage

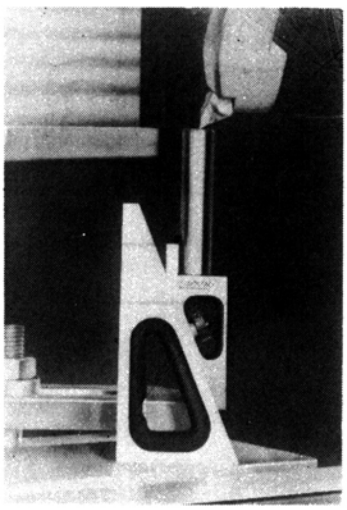


Fig. 513. Use of Planer Gage

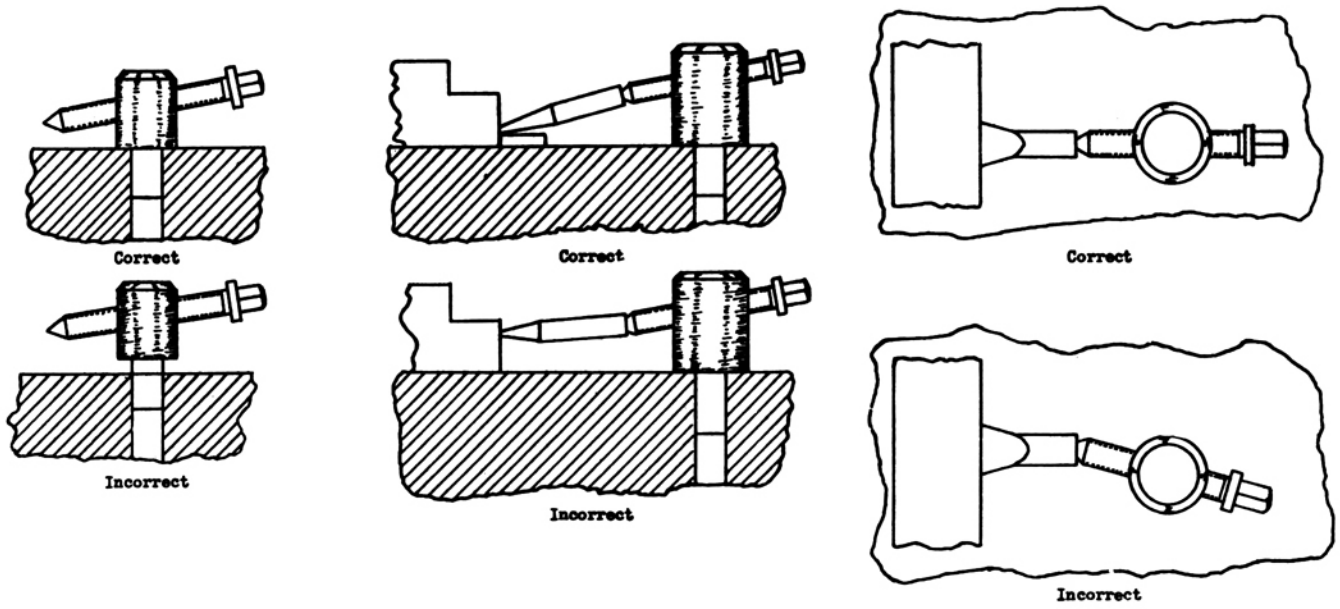


Fig. 514. Correct and Incorrect Use of Toe-Dogs and Poppets

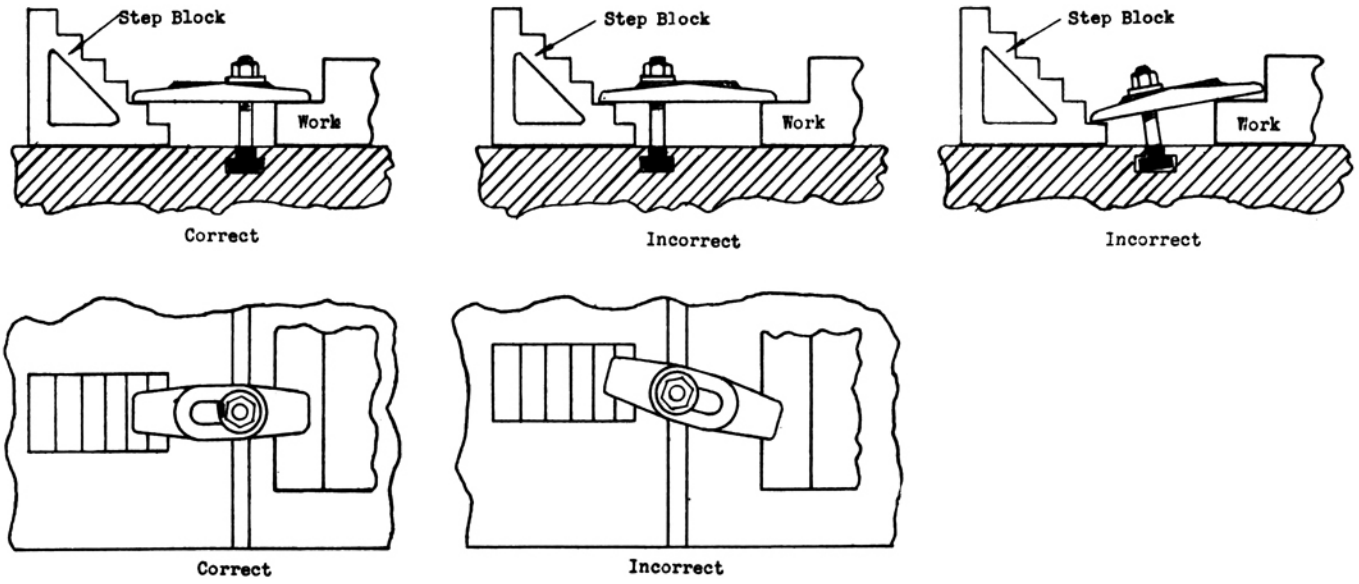


Fig. 515. Correct and Incorrect Methods of Clamping

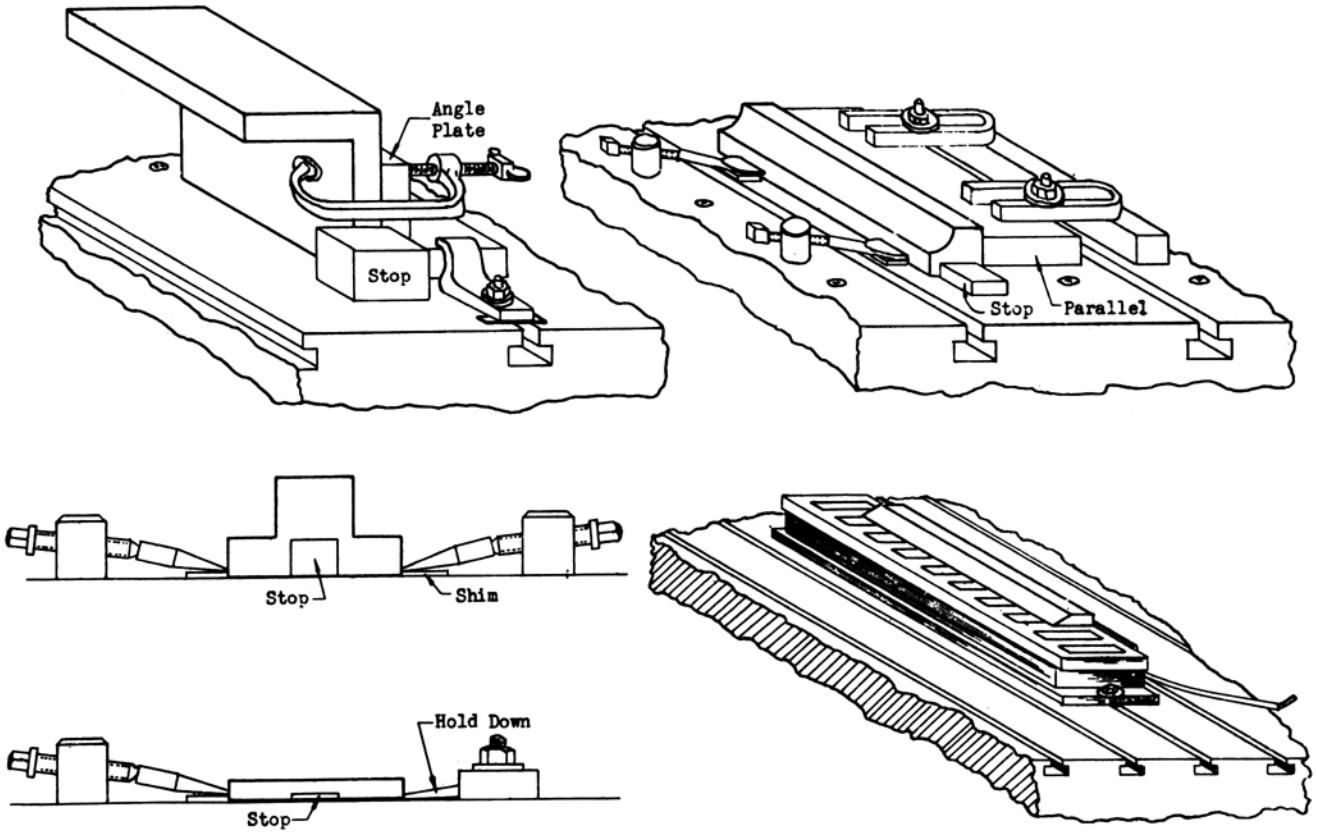


Fig. 516. Planer Set-Ups

Chapter 19

LATHE

The lathe is one of the most important machines in the machine shop. It removes material from revolving work by using suitably formed cutting tools of hardened and tempered steel or of alloy metals such as Carboloy, Stellite, etc.

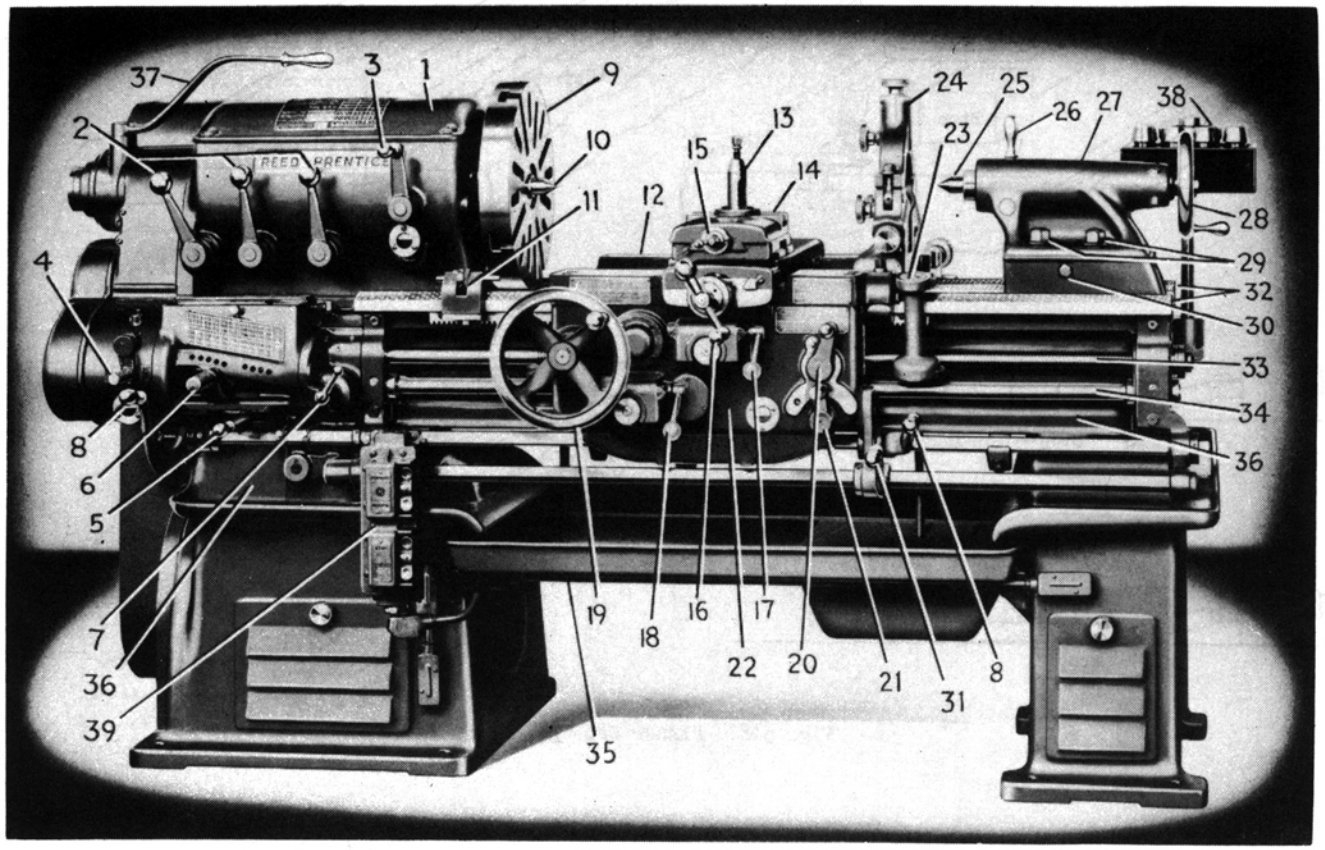


Fig. 517

1. Headstock
2. Speed Change Levers
3. Lever which controls back gears in headstock
4. Lever for controlling series of feed change
5. 2nd Lever for " " " " "
6. Sliding Tumbler Gear Lever
7. Lever for controlling feed to lead screw or feed rod
8. Spindle Start, Stop and Brake Lever
9. Face Plate
10. Live Center
11. Micrometer Carriage Stop
12. Saddle or Carriage
13. Tool Post
14. Compound Rest
15. Compound Rest Feed Screw Handle
16. Handle for Operating Cross Feed by Hand
17. Lever for Controlling Power Longitudinal Feed Clutch
18. Lever for " " Cross Feed Clutch
19. Hand Wheel for Longitudinal Carriage Travel
20. Lead Screw Split Nut Lever
21. Feed Rod and Lead Screw Safety Lever (Prevents engaging both feeds)
22. Apron
23. Thread Chasing Dial
24. Steady Rest
25. Dead Center
26. Tailstock Spindle Binding Lever
27. Tailstock
28. Tailstock Handwheel
29. Tailstock Clamping Screws
30. Screw for "Setting Over" Tailstock (For Taper Turning)
31. Lead Screw Reverse Lever
32. Bed Ways
33. Lead Screw
34. Feed Rod
35. Chip and Oil Pan
36. Bed
37. Collet Attachment Lever
38. Collet Cabinet
39. Electric Motor Control Switch

1. What are the different types of lathes?

A. The engine lathe shown in Fig. 517 is used mostly in tool rooms. Because of its adaptability to various kinds of work it requires a great degree of skill to operate.

The vertical turret lathe is the fastest lathe for short or heavy work.

The horizontal turret lathe, commonly called the screw machine, is used extensively in the production of duplicate parts. Most of these machines are equipped with a pump and a metal basin so that a lubricant or coolant can be used.

The production lathe and the automatic screw machine are used for the production of small duplicate parts.

2. How is the size of the engine lathe determined?

A. Engine lathes vary in size, ranging from the small bench lathe of only a few inches to one many feet in length. The size of the engine lathe is based upon two measurements--the approximate largest diameter that can be revolved over the ways and the total length of the bed. The actual size of the maximum swing is usually somewhat greater than the nominal size listed. For example, an 18 inch lathe may actually swing $18\frac{1}{2}$ inches over the ways of the bed.

3. Name six important parts of the lathe.

A. Bed, carriage, headstock, tailstock, thread cutting mechanism, and feeding mechanism.

4. Describe the bed.

A. The bed consists of two heavy metal sides located lengthwise, with ways or V's formed upon them. The bed is rigidly supported by cross-girths.

5. What is the carriage and what is its function?

A. The carriage, consisting of the saddle and apron, is the movable part which slides over the ways between the headstock and tailstock. The saddle has the form of the letter H, being bridged across the lathe bed to carry the cross-slide and tool rest, fitted to the outside ways, and gibbed to the bed. The apron (shown in the two illustrations of Fig. 518) contains the gears and clutches for transmitting motion from the feed rod to the carriage and the split nut which engages with the lead screw in cutting threads. The function of the carriage is to carry the compound rest.

6. Describe the headstock.

A. The complete headstock (Fig. 519) consists of the headstock casting, which is located on the ways at the left of the operator, the spindle in which the live center is

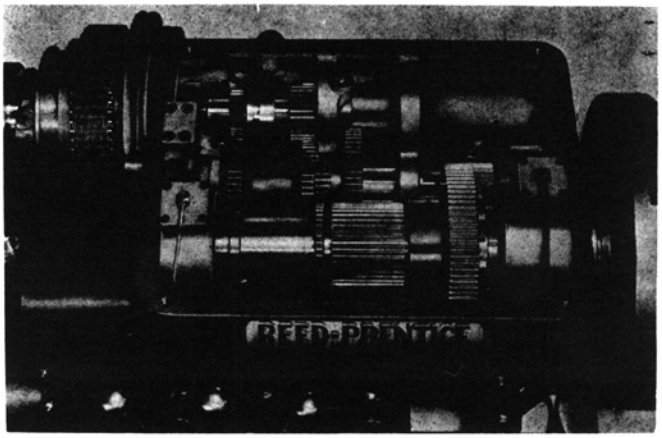
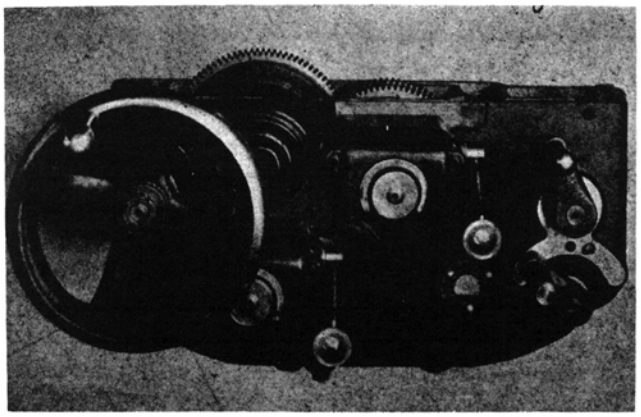
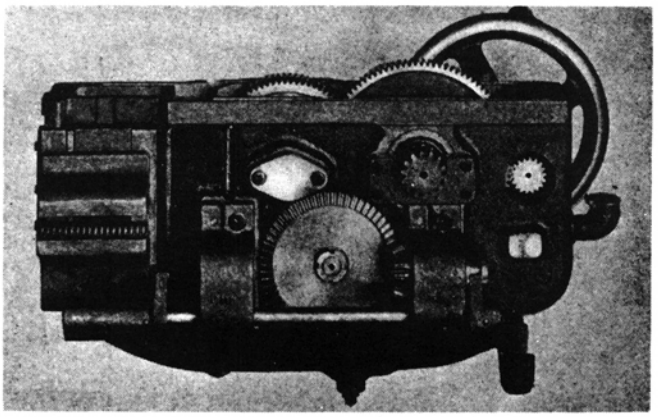


Fig. 519



(a)



(b)

Fig. 518. Apron: (a) front, (b) rear.

rigidly held by a taper, and the necessary gears and mechanism for obtaining the various spindle speeds.

16 Speed Index Plate

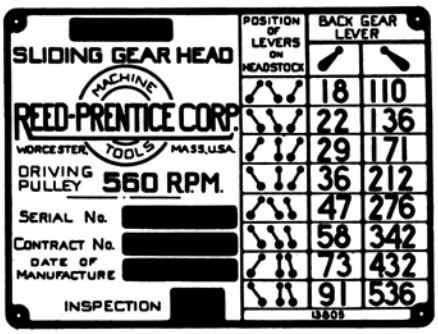


Fig. 520. Typical instruction plate for engaging gears in headstock.

7. Describe the tailstock.

A. The tailstock (Fig. 521) is a movable casting located opposite the headstock on the ways. It contains the dead center, the adjusting screw, and the hand wheel.

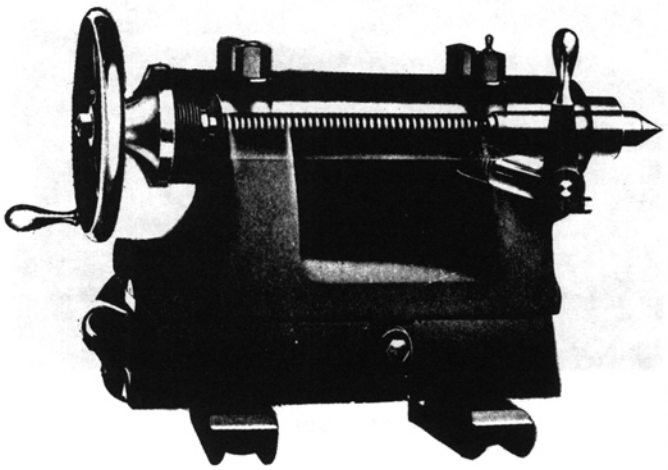


Fig. 521

8. What are the thread cutting and feeding mechanisms?

A. The thread cutting mechanism includes any gears or mechanism which transmits motion from the main spindle to the lead screw. The carriage movement is adjusted by the split nut on the lead screw, using the lever on the outside of the apron to move the split nut (see Fig. 518). The lead screw is used only for thread cutting, except on a lathe that does not have a feed rod. The apron on such a lathe is moved along by a spline in the lead screw instead of by using the thread.

The feeding mechanism is a train of gears (series of gears in mesh) which transmits motion from the headstock or main spindle to the feed rod, and also to the lead screw when it is being used. The motion is then transmitted from the feed rod to various gears in the apron. The feed gears are generally controlled by friction, through small knobs located on the front of the apron. Fig. 522 shows the quick change gears used in securing the correct number of threads per inch in thread cutting or for varying the feed to suit different conditions. The plate (Fig. 523) indicates the position of the levers for producing the desired feed in thousandths or threads per inch.

9. What is the purpose of the compound rest?

A. The compound rest supports the cutting tool in its various positions. It may be swiveled on the cross-slide to any angle in the horizontal plane. Its base is graduated to 180°, or 90° each way from the center, and the feed screw collar is graduated in thousandths of an inch. A compound rest is essential in turning and boring short taper work and in turning angles and forms on forming tools.

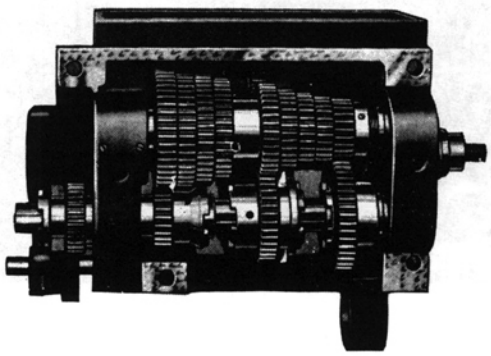


Fig. 522

THREADS PER INCH										END GEARS
48	52	56	64	72	C	80	88	92	96	COMPOUNDED
24	26	28	32	36	B	40	44	46	48	
12	13	14	16	18	C	20	22	23	24	DIRECT
6	6½	7	8	9	B	10	11	11½	12	
3	3¼	3½	4	4½	A	5	5½	5¾	6	
FEEDS IN THOUSANDTHS										
7	6½	6	5¼	4¾	C	4¼	3	3.6	3½	COMPOUNDED
14	13	12	10½	9½	B	8½	7¾	7½	7	
28	26	24	21	19	C	17	15½	14¾	14	DIRECT
56	52	48	42	37½	B	33¾	31	29½	28	
112	104	96	84	75	A	67½	61½	59	56	

Fig. 523

In cutting American National and Sharp V threads the compound rest is swiveled to 30° from the center, which is one-half the included angle of these threads. The reason for this is that the cutting tool acts similarly to a facing tool when it is forced into the work at an angle of 30° instead of 90°. This causes the tool to cut only on one side, giving a better finish to the thread because the chips do not have a tendency to clog up the point.

CAUTION. Take great pains not to force the corner of the compound rest into the lathe chuck. This is done too frequently. Always keep the dovetail slide covered with the compound rest, to exclude all dirt and protect the slide from being nicked or marred by tools falling on it.

When facing the thickness of a piece of work, such as a spacer, to a size which is held to close limits, the compound rest may be swiveled at an angle of 30° from the face of the work and the tool fed in with the compound feed screw. The distance this screw advances is shown on a dial graduated in thousandths. Fig. 524 shows the compound rest set at a 30° angle. The tool is fed in along the hypotenuse of triangle ABC. From a theorem in geometry (which states that in a right triangle the side opposite the 30° angle is equal to one-half the hypotenuse) it can be seen that the thickness of the work is reduced one-half thousandth for every thousandth the tool is advanced with the compound feed screw.

In a 30° right triangle the side opposite the 30° angle is always one-half the hypotenuse. In Fig. 524 BC equals one-half of AB. Then if point B is moved in .002" along the line AB, BC will move in .001" against the face.

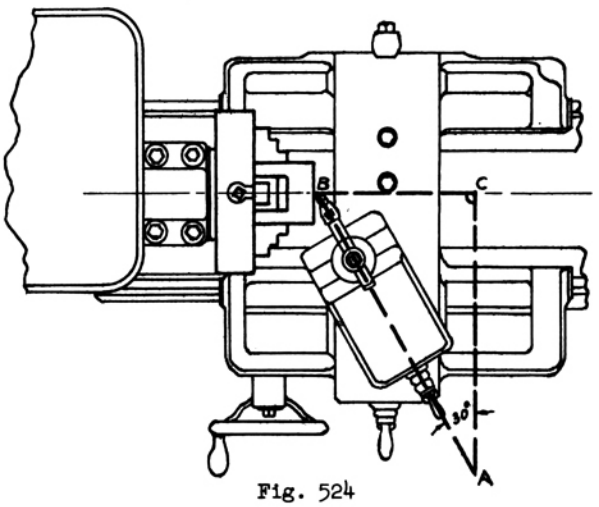


Fig. 524

10. Name and describe attachments used on the lathe.

A. Taper attachment, relieving or backing off attachment, milling attachment, steady rest, and follower rest.

The taper attachment is used to turn and bore tapers. This is considered to be the best method of machining tapers. Fig. 525 shows a taper attachment with guide bar set in position to machine a taper.

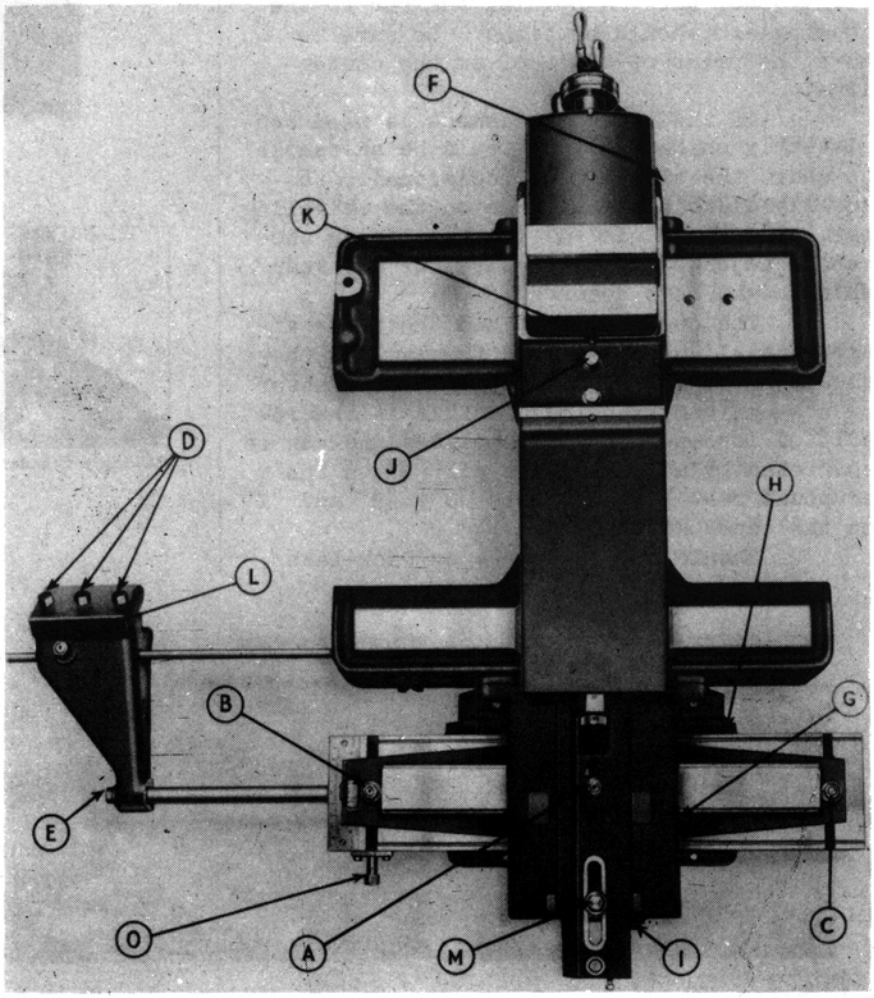


Fig. 525
Saddle with Taper Attachment

The guide bar is graduated in degrees at one end and in taper per foot at the other end.

INSTRUCTIONS FOR OPERATING TAPER ATTACHMENT

To set this attachment for any desired taper, loosen the nut "A" (see Fig. 525) holding the guide-plate to the sliding shoe, and nuts "B" and "C" at the ends of the swiveling bar. Set the swiveling bar to the desired taper indicated on the scale, using the micrometer adjusting screw, and tighten the nuts mentioned above. Except when setting the taper, these nuts should always be tight even when the taper attachment is not being used. Clamp screw "M" and lock nut "L" must be loose when using taper attachment and tightened only when attachment is not in use. To engage or disengage the taper attachment, tighten or loosen the nuts marked "D" which clamp the locking arm to the bed. When the taper attachment is being used, nut "E" holding the bar to the locking-arm should be tight. Be sure to have the point of the tool on the center-line.

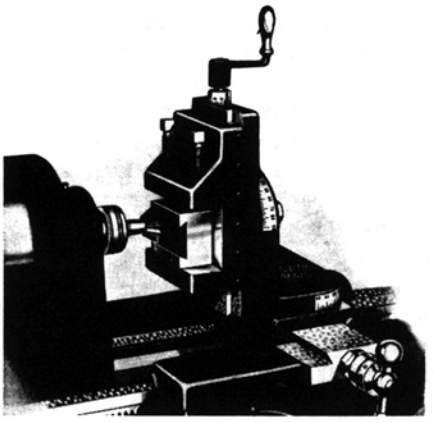
If the taper attachment is used consistently on one job, it would be advisable to shift the sliding bar occasionally to equally distribute any wear on the swiveling bar. The taper gib "G" in the sliding shoe can be adjusted to compensate for any wear which might have occurred.

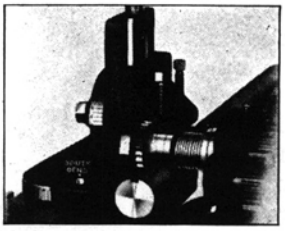
The compound rest and the taper attachment slide should move freely, but there should be no looseness or play. If chatter or non-uniform taper occurs this is the result of looseness in these slides and can be corrected by adjusting the gib "F" on the compound rest base and gibs "G", "H" and "I" on the taper attachment.

Should any looseness or back-lash develop in the cross feed screw nuts it can be removed by loosening screw "J" and tightening screw "K", then retightening screw "J".

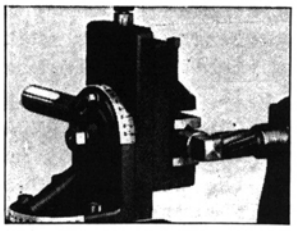
The relieving or backing-off attachment shown in Fig. 526 is used for backing off cutters, taps, etc. It requires very little time to attach to a lathe and the attachment can be removed when the required tools are relieved or backed off.

The milling attachment shown in Fig. 527 illustrates how mill work can be done on a lathe and shows the operations in milling a



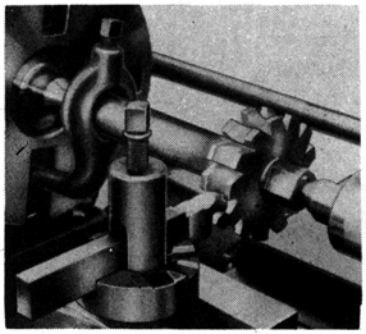


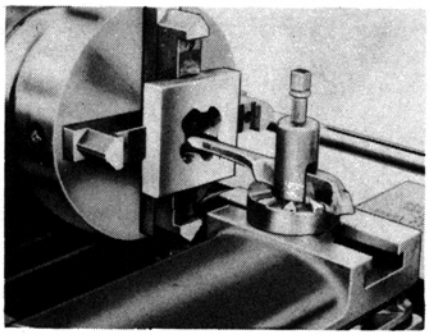
Milling a Standard Keyway.

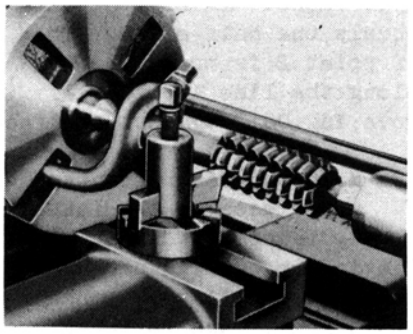


Squaring the End of a Shaft.

Fig. 527







(a) (b) (c)

Fig. 526. (a) Form relieving; (b) internal relieving; (c) helical (or spiral) flute relieving.

dovetail, milling a square end on a shaft, and cutting a keyway.

Fig. 528 shows the steady rest used for turning long shafts of small diameter

Fig. 528

and for boring and threading spindles. It is necessary to support long work while machining it, to prevent taper caused by springing of the work.

The follower steady rest shown in Fig. 529 is attached to the saddle of the lathe for machining work of small diameter that is likely to spring if it has no support. The adjustable jaws of the follower rest bear directly on the finished diameter of the work, following the cutting edge of the tool on the opposite side of the work. As the tool feeds along the work the follower rest, being attached to the saddle, travels with it.

Fig. 529

11. Name the two most commonly used lathe chucks and give the use of each.
    A. The four jaw independent lathe chuck, Fig. 530, is used for holding most of the work that should be held in a chuck. The hardened steel jaws are reversible and will hold work of different sizes and shapes. Each jaw is independent of the other.

The three jaw universal chuck in Fig. 531 holds cylindrical work. All three jaws move universally to bring the work on center. This chuck is easier to operate than the four jaw chuck but it can be used only on round or hexagon stock.

12. Name two plates that are used extensively on the lathe.
    A. The driving plate in Fig. 532 is used

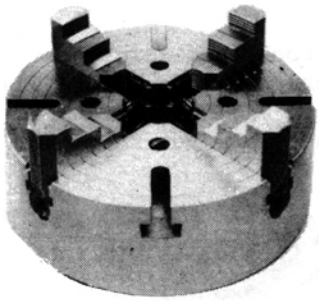


Fig. 530

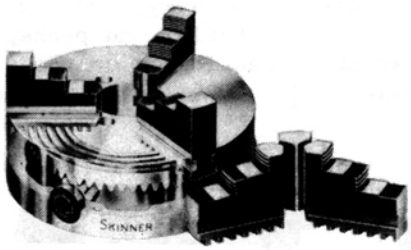


Fig. 531

Fig. 532

Fig. 533

Fig. 534

Fig. 535

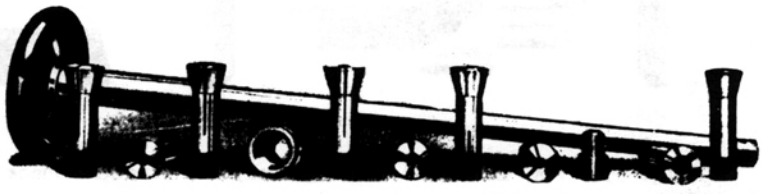


Fig. 536

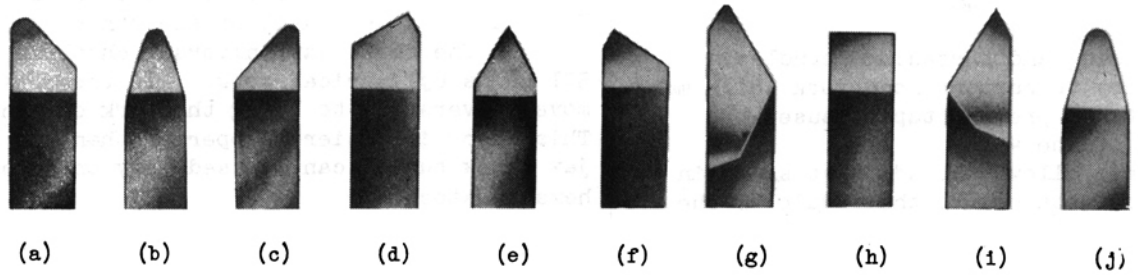
for driving work held between centers. The face plate, Fig. 533, is used to hold work while it is being machined.

13. Name two lathe dogs.

A. Fig. 534 shows a bent tail lathe dog used for driving cylindrical pieces. The clamp dog in Fig. 535 is used for work with flat sides.

14. What attachment should be used for holding cylindrical shafts of small diameter?

A. The draw-in collet chuck attachment in Fig. 536 is used on the lathe for the production of small metal parts and in the tool room for fine accurate work. Long rods are passed through the draw bar and held in the collet chuck while they are being machined. Short work may also be held in this chuck.



(a) (b) (c) (d) (e) (f) (g) (h) (i) (j)

Fig. 537. (a) Left hand turning tool; (b) round nose turning tool; (c) right hand turning tool; (d) left hand corner tool; (e) threading tool; (f) right hand corner tool; (g) left hand side tool; (h) square nose tool; (i) right hand side tool; (j) brass tool.

15. What are some of the most important cutting tools and how are they held?

A. Fig. 537 shows some of the different types of tools used in lathe work. These tools are held in tool holders, some of which are shown in Fig. 538.

16. Tell how the tool holder should be placed in the tool post for turning work.

A. As a general rule place the tool holder in the tool post at approximately 90° with the center line or a little in the direction of the dead center when feeding toward the headstock. The point of the cutting tool should be on the center line between the dead center and the live center, and should not extend far from the tool holder. Neither should the tool holder extend far from the tool post if chattering is to be avoided. A very good illustration is shown in Fig. 539.

CAUTION. When taking a heavy cut do not have the tool pointing in the direction of the live center. If the tool is pointed in that direction and runs into a hard spot the material will have a tendency to push the tool away, forcing the tool to dig into the material and perhaps scrapping the job. This caution is unnecessary in taking a light cut.

Bear in mind that when you force the

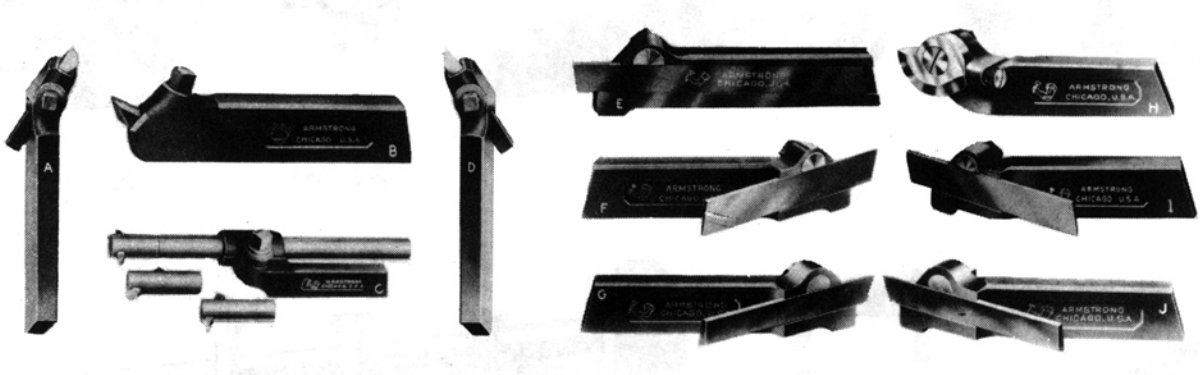


Fig. 538. (a) Left hand turning tool; (b) straight-shank turning tool; (c) boring tool; (d) right hand turning tool; (e) straight shank cut-off tool; (f) left hand off-set cut-off tool; (g) left hand off-set side tool; (h) threading tool; (i) right hand off-set cut-off tool; (j) right hand off-set side tool.

tool into the work .001" on the periphery you are taking .002" off the diameter.

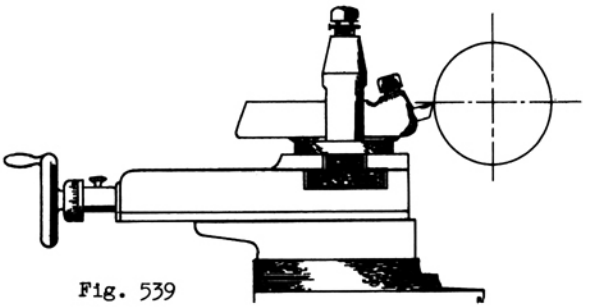


Fig. 539

17. Describe the difference between the dead center and the live center.

A. The dead center is supported in the tailstock and does not revolve. The live center is contained in the spindle and revolves with the work.

CAUTION. Keep the dead center well lubricated with oil.

18. When is the combined drill and countersink used?

A. To machine a piece of stock on centers, each end must be countersunk. This is accomplished with a combined drill and countersink (shown in Fig. 540). The included

Fig. 540

angle of the combined drill and countersink is 60°. This angle is the same as that of the dead and live centers.

19. Why should a countersunk hole be used for a center hole instead of a large prick punch mark?

A. If a large prick punch mark is used instead of a hole made by a combined drill and countersink, the work will ride on the point of the steel center and not on the sides as it should. The work will soon heat and run untrue, due to improper location and lack of lubrication. The hole produced by the combined drill and countersink should contain center compound to lubricate the point of the dead center.

20. What extra care should be taken in inserting a centered piece of work between centers?

A. Do not force the dead center against the work close to the countersunk hole, as that causes the hole to be burred or nicked and the work will not run true.

21. How is the work mounted on centers?

A. When mounting work between centers be careful to see that the centers are in good condition. The dead center in the tailstock will be the first to show wear. As the live center in the spindle revolves with the work, the dead center must be lubricated. Incorrect mounting on centers is shown in Fig. 541. A lathe dog that will not clear

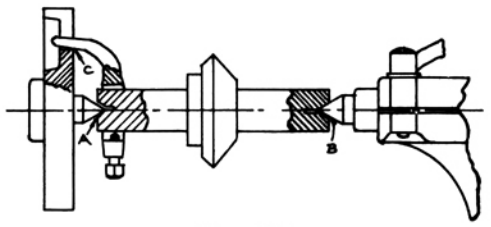


Fig. 541

the bottom of the slot in the driving plate has been used (shown at C). A and B show incorrect bearings which cause the work to revolve eccentrically.

Fig. 542 shows correct mounting between centers. The lathe dog is fastened to

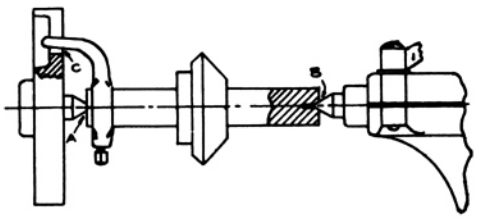


Fig. 542

the work and the tail should clear the bottom of the slot. The work is held firmly but not too tightly on the live and dead centers. Adjusting the centers to the work can be learned in a short time. Too great a pressure of the dead center against the work will cause a squeaking noise and the center and work will heat up so as to cause wear from the friction.

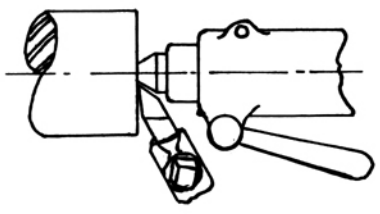


Fig. 543

22. How should the facing or squaring tool be set and used?

A. Fig. 543 shows a cylindrical piece of work being faced off to true the ends. The tool is pointing toward the live center and is not at right angles with the center line. Only light cuts are taken to true the

work. Use a steady rest and chuck if you wish to remove a large amount of stock on the end of a shaft.

23. What is the first thing to do when given a piece of work, and a drawing which gives several length dimensions and the overall length?

A. Add the several lengths and see if they coincide with the overall length given. Also determine whether the work has sufficient stock to finish to the O D shown on the drawing.

24. Give several reasons why the end of a piece of work may not be 90° with the center line when faced with a side tool.

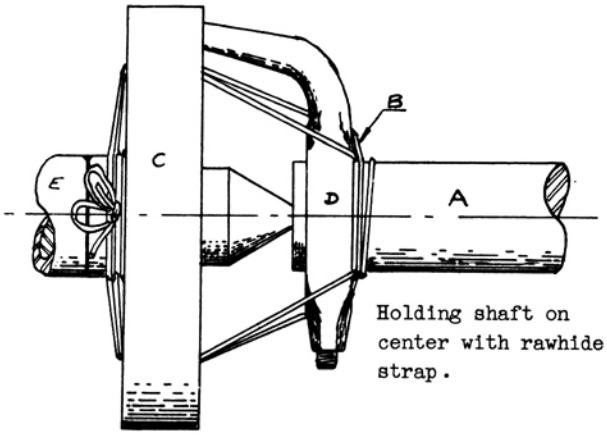
A. The lathe centers may be out of line, the tool may be springing away from the work, the cutting edge of the tool may have worn off, or the gibs of the compound rest may be loose.

25. When using a driving plate and a steady rest, what is the best method for holding a shaft on center so that the end of the shaft can be bored?

A. The best way to do this job is to tie it to the driving plate as shown in Fig. 544. The plate is unscrewed about three or four revolutions from the shoulder of the spindle. Then the work is placed on the center and tied securely to the driving plate with a rawhide strap. Finally the driving plate is screwed to the shoulder of the spindle. This tightens the strap on the work and holds it firmly.

26. Name some operations performed on the lathe.

A. Straight (cylindrical) turning, taper turning (by using the compound tool rest or taper attachment), facing (cutting at right angles to the axis of the work), either



Holding shaft on center with rawhide strap.

Fig. 544

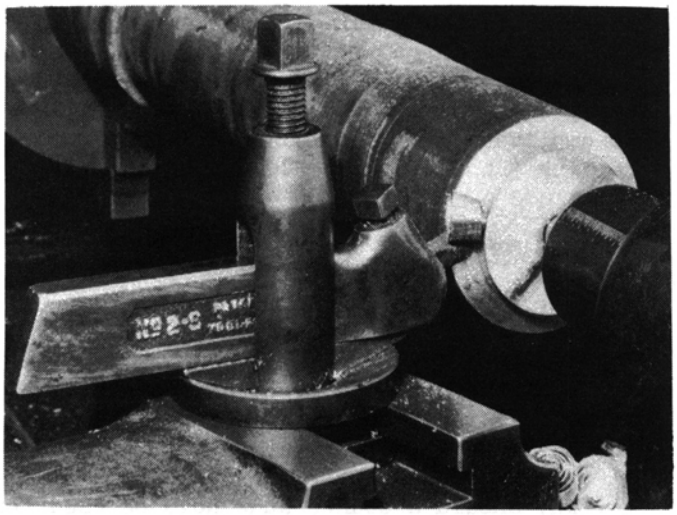


Fig. 545. Turning the O D.

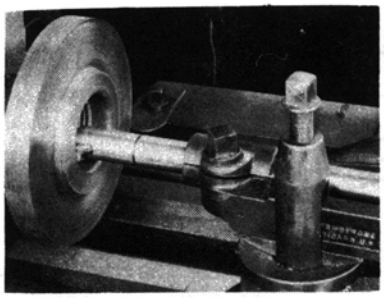


Fig. 546. Machining Radius with Boring Tool.

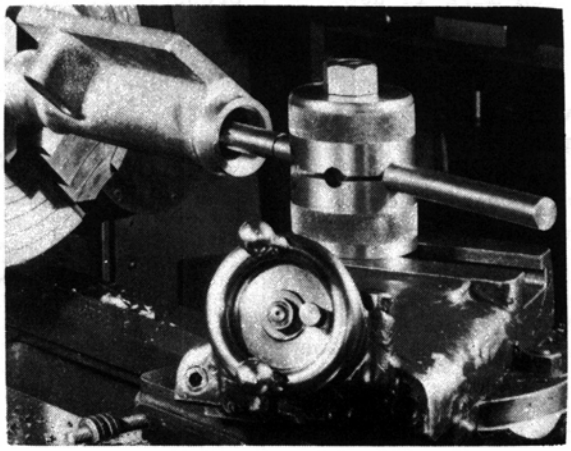


Fig. 547. Internal Threading using Boring Bar.

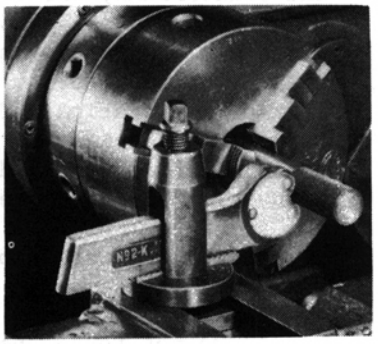


Fig. 548. Knurling.

straight or taper boring, knurling, and external and internal thread cutting. A thread may be chased with a tap or die or cut with a threading tool.

Some lathe operations are shown in Figs. 545 to 551.

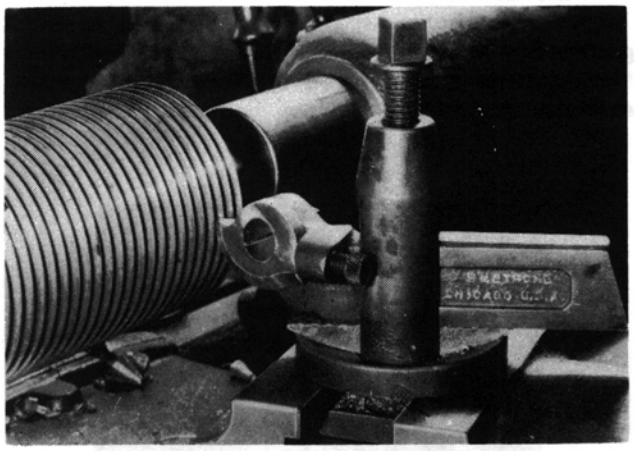


Fig. 549. Cutting a National Standard Thread.

27. What is the difference between an arbor and a mandrel?

A. An arbor is a shaft used for holding and driving cutting tools. Usually one end of the arbor is tapered to fit the tapered hole in the spindle of the machine.

A mandrel is a shaft or bar for holding work to be machined. Plain mandrels are usually tapered .001 to .005 of an inch. Expanding mandrels are made in many types and sizes.

28. How much grinding stock should be left on a shaft 1" in diameter and 48" long if it is to be ground soft?

A. Leave .015 of an inch.

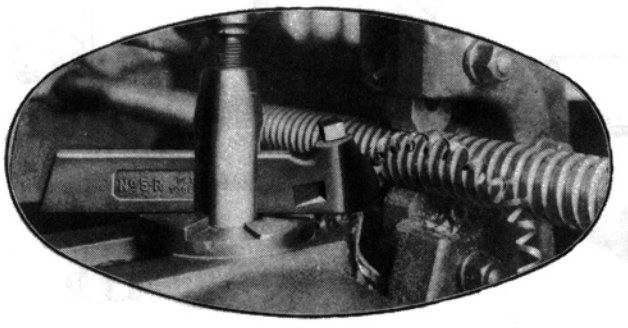


Fig. 551. Cutting a square thread on work which is being supported by a follower rest.

29. How much grinding stock should be allowed on a shaft 1" in diameter and 48" long if it is to be hardened and ground?

A. Leave .035 of an inch.

30. How much grinding stock should be left on the O D and I D of a bushing with 2" bore and $2\frac{1}{2}$" O D if it is to be ground soft?

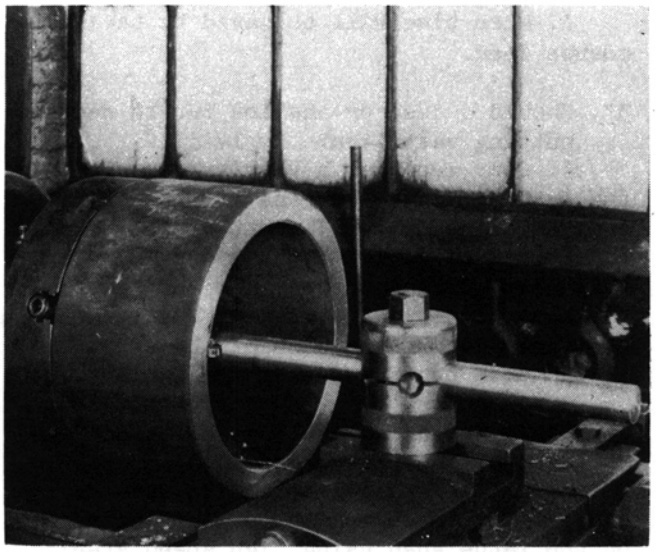


Fig. 550. Boring With A Boring Bar.

A. Leave .015 of an inch on each diameter.

31. How much grinding stock should be allowed on the bushing in question 30 if it is to be made of high speed steel, hardened and ground?

A. Leave .025 of an inch.

32. How may cast iron be machined rapidly and efficiently?

A. Use soda water with a slow speed and heavy feed.

33. How should a tool be ground for finishing cast iron?

A. Grind the tool with a flat top, rounded nose, and with small side clearance.

34. When a drawing calls for grinding on a diameter and against a shoulder, what kind of an under-cut should be given?

A. The under-cut should be made with a narrow, round nose tool fed in at an angle of 45°.

35. What kind of a tool is used to under-cut corners on a job that is to be hardened and ground?

A. Use a round nose tool, as a square nose tool will have a tendency to cause the steel to crack or break when it is hardened.

36. Why is it a good policy to use a coarse feed when machining stock that is to be ground?

A. Much time will be saved by taking a coarse feed.

37. Should a deep or shallow cut be made in cutting cast iron? Explain.

A. Cast iron has a very hard scale on the outside, caused by the iron becoming chilled in casting. In cutting this iron the tool must be immediately forced under this scale or the cutting edge of the tool will be ruined. Never take a shallow cut on cast iron unless the size of the stock is almost down to the size of the finished job.

38. State the usual cutting speeds for the most commonly used metals.

A. The peripheral speeds in feet per minute (fpm) recommended for cutting metals on the lathe when using high speed steel tool bits are shown in the chart below.

MATERIAL	FPM
Chrome, Non-Shrink, H S Steel.....	40 - 50
Cast Iron.....	55 - 65
"RR" (Tool) Steel.....	60 - 65
"AAA" Steel.....	80 - 85
Cold Rolled Steel, Machine Steel..	110 - 120
Brass.....	115 - 125
Bronze, Copper, Aluminum.....	200 - 250

39. What precaution should be taken in fitting a plug gage into work that has just been bored?

A. Before fitting a plug gage into work be sure that the temperature of the work is almost the same as that of the gage or the plug gage may freeze in the work.

40. Name four methods of producing tapers on the lathe.

A. For short tapers the compound rest is generally swiveled to the required angle and the cutting tool is fed by hand. The taper attachment, Fig. 525, may be used to produce

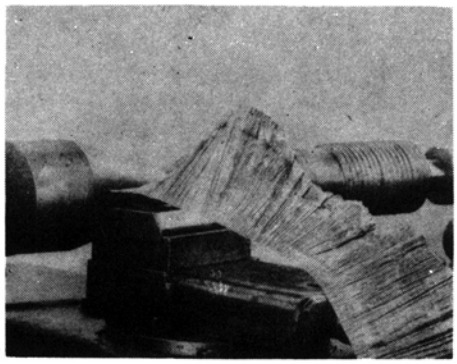


Fig. 552

internal and external tapers. The tool is fed automatically. The third method used to produce a taper is to off-set the tailstock. The tailstock, Fig. 521, has set screws on the side which can be adjusted. The fourth method, producing a taper with a square nose tool, is illustrated in Fig. 552.

CAUTION. In producing tapers the

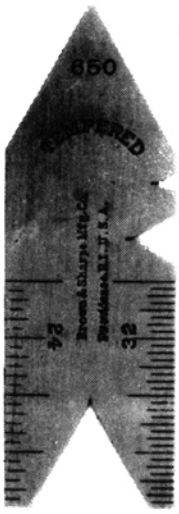


Fig. 553

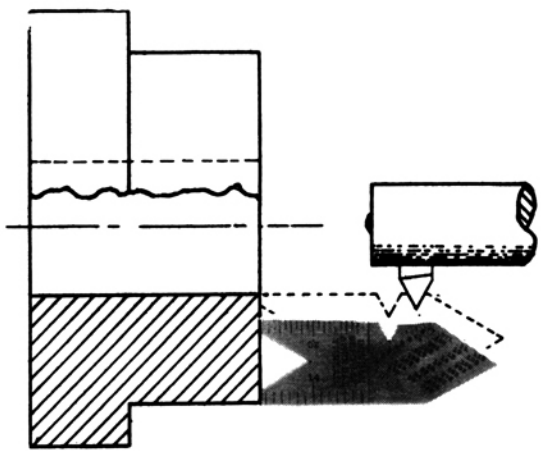


Fig. 554

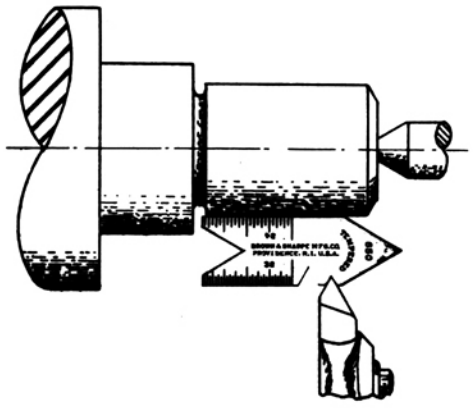


Fig. 555

point of the tool must be on the center line of the work.

41. What method should be used to produce a short taper having an included angle of 60°, such as a lathe center?

A. This taper may be produced by using either the compound rest or the square nose tool. The method which uses the compound rest is considered the better of the two.

42. What method should be used in producing a long shaft having a taper of .015" per foot?

A. Use the taper attachment for this type of job.

43. Name several ways of checking tapers.

A. A taper may be checked by a micrometer, by measuring the large and small diameters, by taper gages, by fitting to the spindle or sleeve for which it is intended, and by using the sine bar and Johansson Gage Blocks.

44. What is the center gage and for what is it used?

A. The center gage is a small flat piece of steel which is graduated in fractions of an inch. Included angles of 60° are cut in it, as shown in Fig. 553. It is used in setting American National and Sharp V threading tools, as shown in Figs. 554 and 555.

45. Explain clearly what is meant by the pitch and lead of a screw thread.

A. The pitch is the distance from a point on one thread to the corresponding point on the next, measured parallel with the axis of the work. The lead is the distance a nut would advance on a screw thread in each revolution. The lead of a single thread screw is equal to the pitch, the lead of a double thread screw is equal to twice the pitch, etc.

46. In setting a threading tool for cutting a tapered thread, as on a pipe, should the center line of the tool be set at 90° with the center line of the work or at 90° with the line of taper?

A. Set the center line of the tool at 90° with the center line of the work.

NOTE. The taper attachment of the lathe should be used in cutting pipe threads or on any other tapered thread job.

47. When a lathe has been set for cutting threads and a trial cut taken on the job, how is it checked for the correct number of threads per inch?

A. Use a rule and count the number of threads per inch, as shown in Fig. 556, or

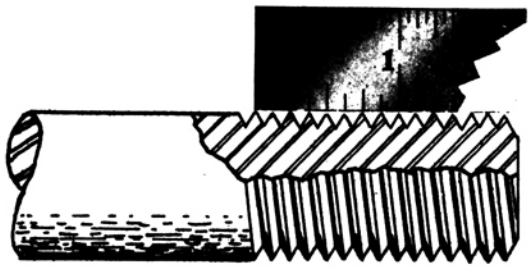


Fig. 556

use a screw pitch gage, as shown in Fig. 557.

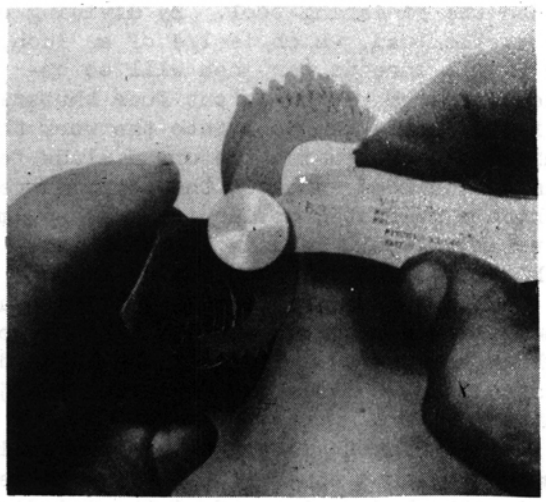


Fig. 557

Major Diameter	Threads Per Inch	Major Diameter	Threads Per Inch
1/4	20	9/16	12
5/16	18	5/8	11
3/8	16	3/4	10
7/16	14	7/8	9
1/2	13	1	8

48. State the number of threads per inch for standard American National Coarse threads.

A. The table above gives the number of threads per inch for the standard American National Coarse thread series (up to one inch in diameter).

49. Explain the difference in cutting a right-hand and a left-hand thread.

A. To cut a right-hand thread the compound rest is set over toward the right on the saddle at the proper angle. The feed of the carriage is toward the headstock. In cutting a left-hand thread the compound rest is moved toward the left on the saddle and the feed of the carriage is toward the tailstock.

50. Describe three methods of cutting a double thread.

A. The following examples will show how to cut a double thread having 1/4" lead and 1/8" pitch:

Swivel the center line of the compound rest to a line parallel with the dead and live centers. In cutting the first groove, use a roughing tool, which is narrower than the finishing tool. By dividing one inch by the lead, which is 1/4 of an inch, four single threads per inch will be obtained. Set the lathe to cut four threads per inch. Force the tool into the work the required depth by using the cross slide feed. By using the feed screw on the compound rest, on which a graduated collar is attached, move the roughing tool over the length of the pitch and proceed to rough out as before.

Another way to do this job is to use a face plate having two equally spaced slots in which to insert the tail of the lathe dog. Be careful not to move the dog after the job is started.

Still another way to do this job is to mark two teeth equally spaced on the gear that is attached on the end of the spindle, and after roughing out the first groove, disengage the marked gear on the spindle and turn both spindle and gear one-half a revolution then mesh. You are then ready to rough out the second groove.

51. Describe knurling.

A. Knurling is a process of making a series of indentations or depressions on the surface of the work, and is done by a knurling tool held in the tool post. Fig. 558 shows a good example of knurling.

A slow speed should be used to knurl a piece of work. Force the knurling tool slowly into the work to a depth of approximately 1/64 of an inch. Set the feed so that the tool will feed across the surface of the work. Reverse the feed of the carriage at the end of the cut, and repeat the operation until the knurling has been completed. The

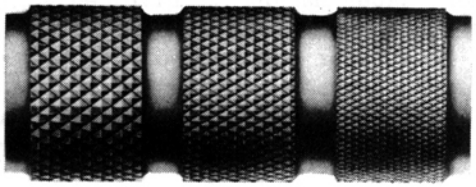


Fig. 558. (Left) coarse, 14 pitch; (center) medium, 21 pitch; (right) fine, 33 pitch.

knurls should be lubricated while cutting. An illustration of a knurling job is shown in Fig. 548.

52. What are some of the safety rules which should be observed in the lathe department?

A. The following safety rules apply particularly to the lathe department. They should be observed by every one working on a lathe.

1. Have the first piece of every job inspected.
2. Do not touch any chips with your hand. Use a wooden paddle.
3. Do not use a Westcott wrench on any nut or screw. Use an open end wrench, a compound wrench, and the standard tool post wrench.
4. Do not leave the chuck wrench in the chuck at any time. Oil the spindle nose before putting on chuck or plate.
5. Be neat. Do not allow oil or chips to collect around your lathe.
6. Keep tools off ways and top of carriage.
7. Cover all boring and internal threading tools with a towel while using gages or calipers to check bore.
8. Goggles must be worn on the following types of work:
   - Turning of square, hexagon, or irregular stock of any kind;
   - Turning of all broach jobs;
   - On bronze or brass at all times;
   - On all tough stock where the chips have a tendency to fly.

Chapter 20

TURRET LATHES

HORIZONTAL UNIVERSAL TURRET LATHE

The screw machine was an early development from the lathe and was primarily designed for the rapid production of screws, as the name implies. This machine gained wide prominence later by the addition of certain tools and attachments which could be used for the completion of small cylindrical duplicate parts.

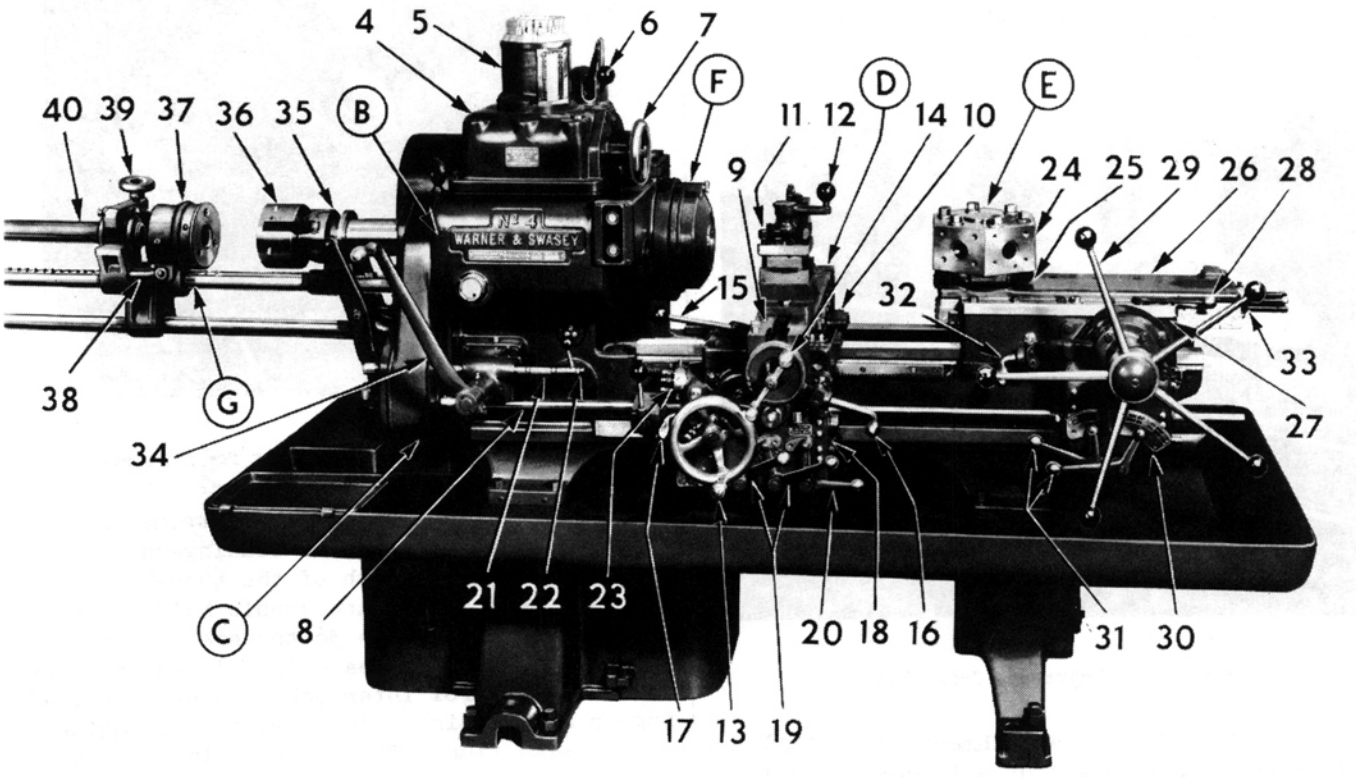


Fig. 559

"B" HEADSTOCK (No. 4 and No. 5 Universal)
4. Surface Speed Preselector
5. Preselector Drum Housing
6. Spindle Control Lever
7. Preselector Handwheel

"C" CHANGE GEAR BOX
(For halving or doubling all feeds)
8. Feed Shaft

"D" UNIVERSAL CROSS SLIDE CARRIAGE UNIT
9. Cross Slide
10. Carriage
11. Square Turret
12. Square Turret Indexing and Binder Handle
13. Longitudinal Feed Handwheel
14. Cross Feed Handwheel
15. Carriage Clamp
16. Cross Feed Engagement Lever
17. Longitudinal Feed Engagement Lever
18. Carriage Apron
19. Feed Shift Levers
20. Feed Reverse Lever
21. Longitudinal Stop Rod
22. Master Stop Screw
23. Longitudinal Feed Stop Roll

"E" HEXAGON TURRET UNIT
24. Hexagon Turret
25. Circumference Binder Ring
26. Turret Slide
27. Saddle
28. Slide Binder Lever
29. Turnstile
30. Saddle Apron
31. Feed Shift Levers
32. Feed Engagement Lever
33. Stop Roll

"F" COLLET CHUCK
34. Bar Chuck Lever
35. Bar Chuck Wedge
36. Finger Holder

"G" RATCHET BAR FEED
37. Bar Feed Head
38. Ratchet Pawl Lever
39. Bar Feed Tube Clamp
40. Bar Feed Tube

A turret, usually of hexagon shape, and a saddle were placed on the machine. The turret held different tools, such as drills, reamers, counterbores, and cutters. The machine was then called a turret lathe by several manufacturers; a horizontal type is shown in Fig. 559. The horizontal turret lathe will be found in most tool rooms, as it can produce small duplicate cylindrical parts in less time than the engine lathe.

The universal carriage (Fig. 560) has both power cross feed and longitudinal feed and may be used for turning, necking, facing, etc., while the hexagon turret is turning and boring. A graduated dial with observation clips gauges the depth of a cut when using the cross feed.

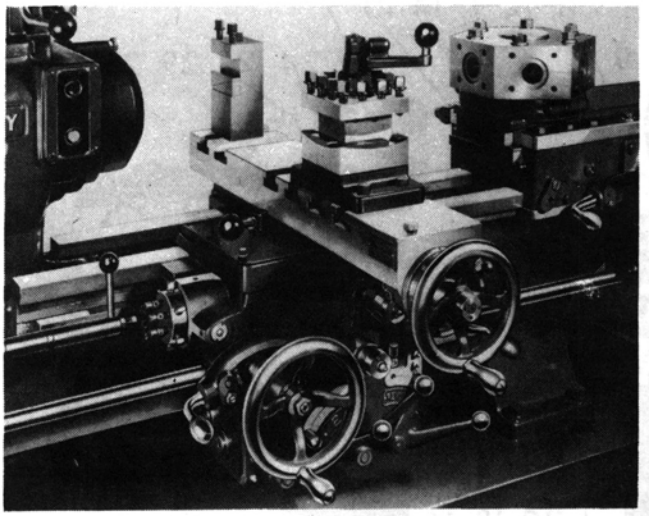


Fig. 560. Universal Cross Slide

The chasing attachment (Fig. 561) cuts threads by means of a leader and follower. The leader is clamped on the carriage feed rod and the follower nut is bolted to the chasing lever. Each leader cuts only one thread.

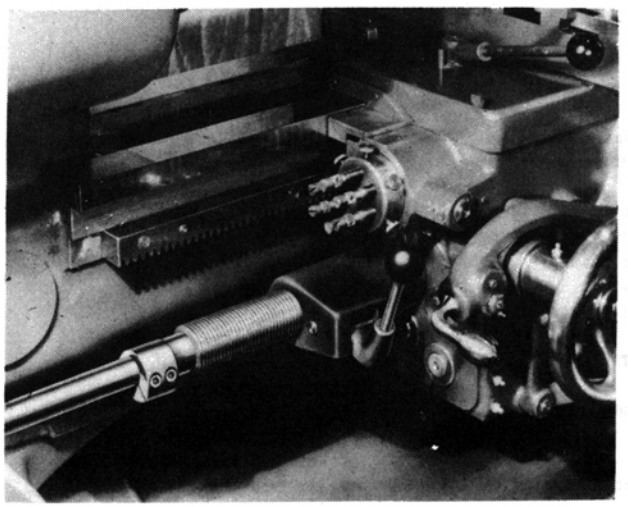


Fig. 561. Chasing Attachment for Carriage

The taper attachment (Fig. 562) is used to turn or bore tapers up to certain sizes and lengths. The tool block of this attachment slides on a base which is fitted to the carriage.

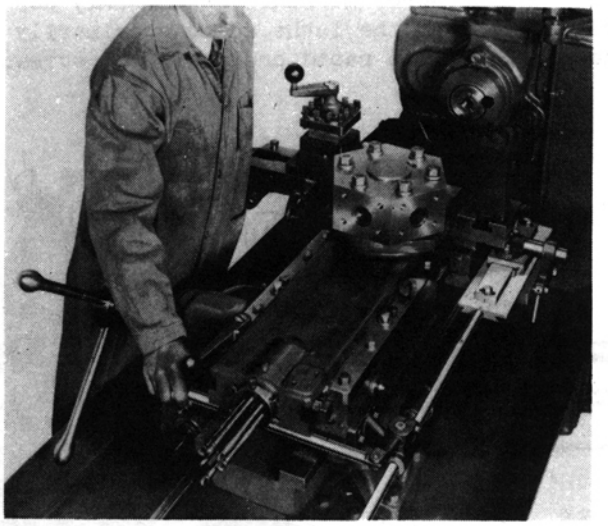


Fig. 562. Taper Attachment for Carriage

The attachment at the extreme right of Fig. 563 offers a means of leading-on taps and dies to the full length of the turret slide stroke. Each leader and double half nut will cut any one pitch 4 to 36 threads per inch when using standard gears in the head end gear box. The control interlock prevents the leading-on and feeding mechanisms from engaging at the same time. The turret slide stop screws

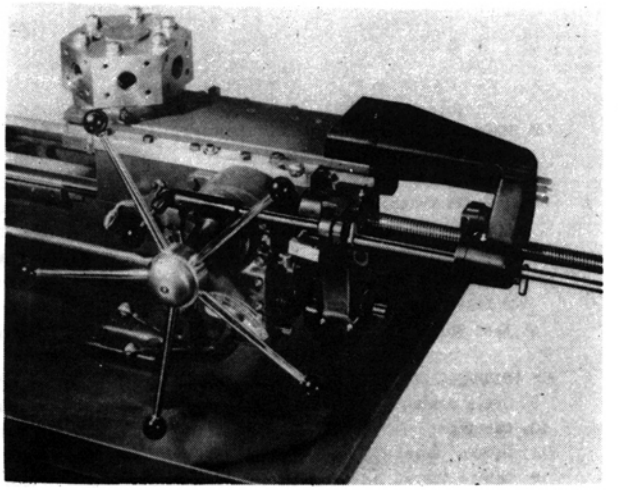


Fig. 563. Leading-on Attachment for Ram Type Turret Lathes

can be set to disengage the single clutch at the end of the thread.

FULL UNIVERSAL THREADING TURRET LATHE

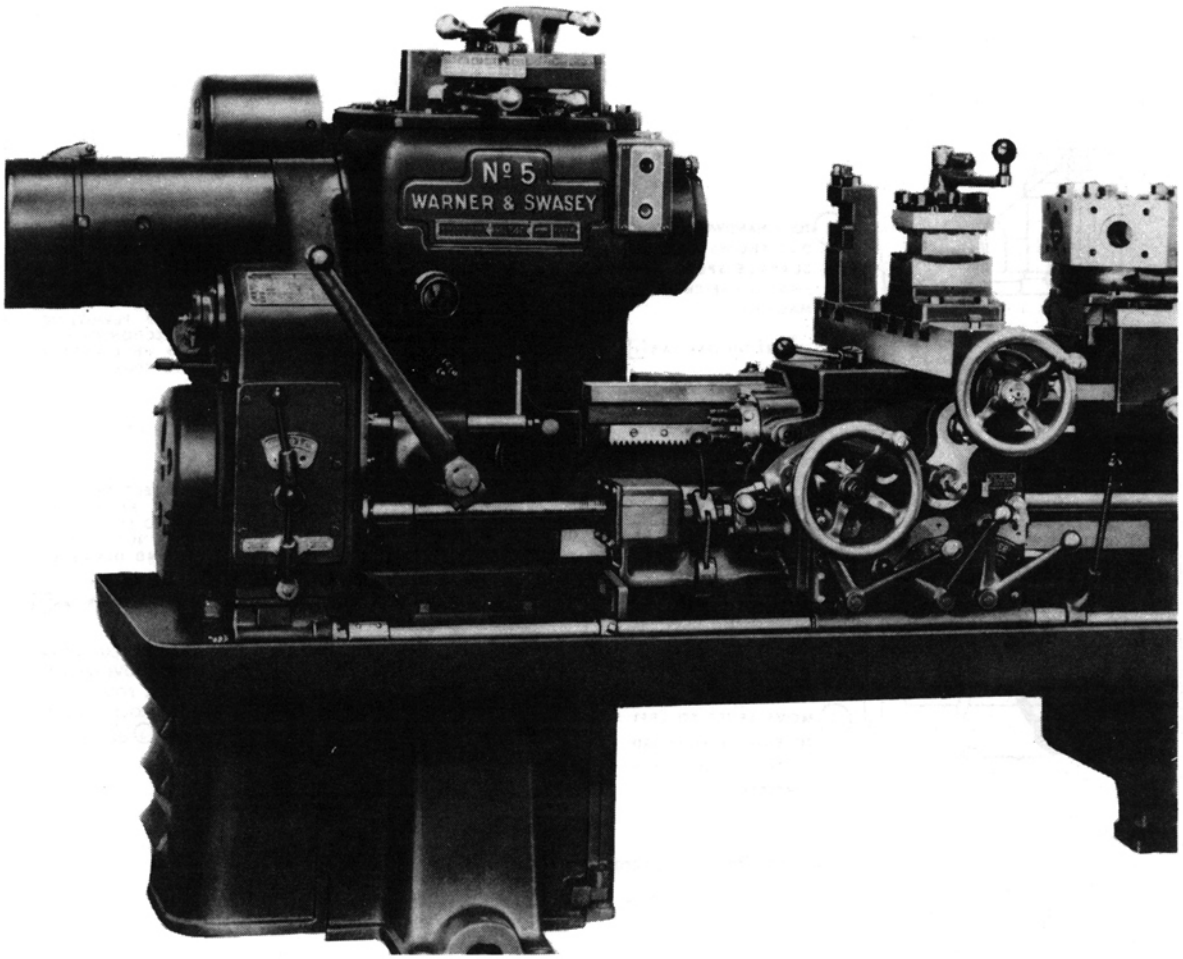
Tool room threading performance, and high speed production advantages are combined in a thread cutting lead screw machine (Fig. 564). The single thread fine pitch lead screw is used for thread cutting only. The remote control lever, located at the operating position, gives forward and reverse motion to the carriage. It is not necessary to disengage the double half nuts or reverse the spindle for the threading operation. This remote control lever is also used in cutting left-hand threads. An automatic knock-off for right-hand threads permits threading to accurate shoulder lengths. The change from threading to regular feeding is accomplished by shifting a single lever on the head end lead screw gear box. A safety interlock prevents the screw cutting and feeding mechanisms from engaging at the same time. The threading range is 2 to 56 threads per inch in lengths to 20 inches.

Operating pre-selector (Fig. 565):

A) Set the pre-selector when setting up tools. First, rotate the drum until the desired feet per minute which show on the dial, correspond to the work diameter to be cut. Then, number that cut, using master numeral (1) at the top of the drum.

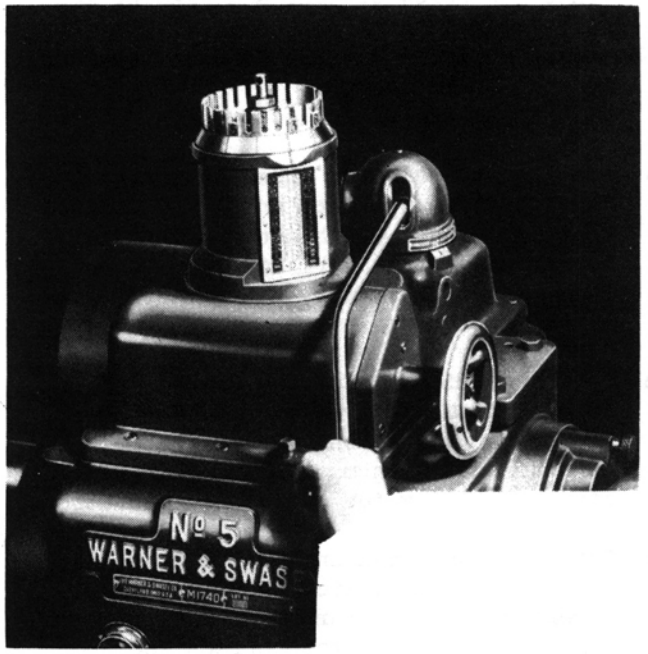
B) Set the indicator for the second cut, by selecting the desired surface feet per minute for the second work diameter. Number this cut with master numeral (2). Continue similarly for additional operations.

C) Cuts are now shown in order by the master numerals 1, 2, 3, etc. Now, use these numerals in their turn from one cut to the next. In this way the correct feet per



Full Universal Threading Turret Lathe.

Fig. 564



Pre-Selector.

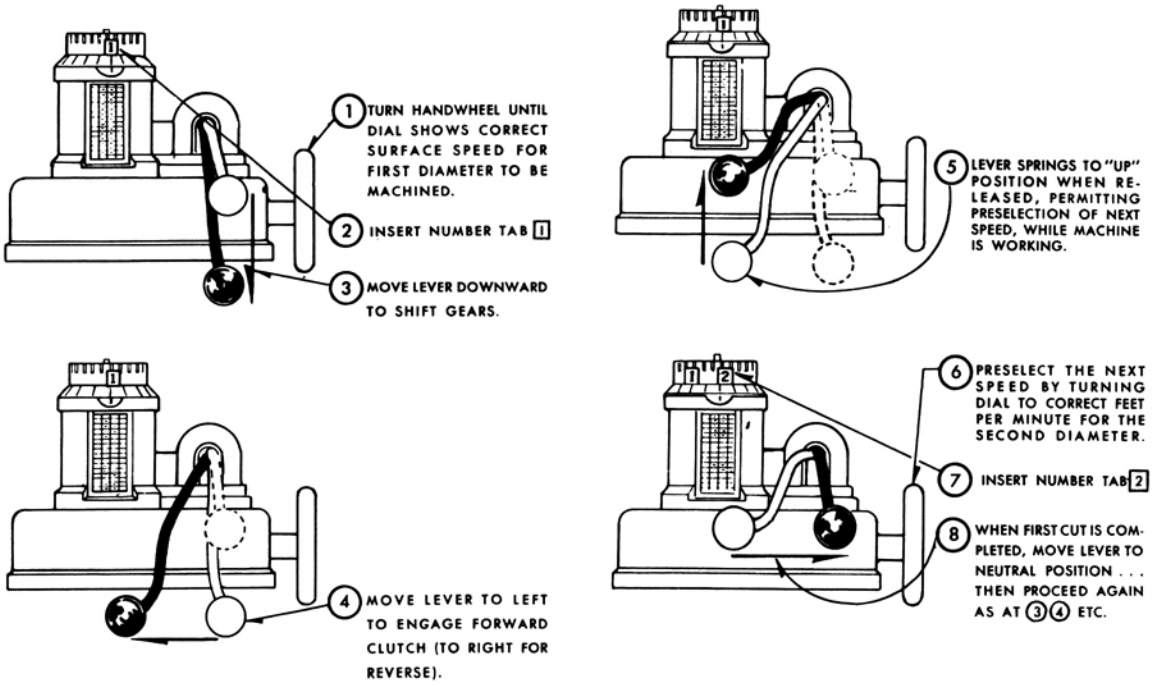
Fig. 565

minute will be set for each cut, regardless of the number of speed changes.

D) In actual use it is possible to pre-select the next speed by turning to the next numeral while the machine is cutting. Then, when the cut is finished it is only necessary to move the lever, bringing the next speed into play, without stopping the spindle. Forward--reverse--stop--start with the one lever.

Pre-selector is set for surface speeds as shown in Fig. 566.

The operator can select the correct spindle r.p.m. to produce the proper surface feet per minute for any diameter to be cut by turning the handwheel.


1 TURN HANDWHEEL UNTIL DIAL SHOWS CORRECT SURFACE SPEED FOR FIRST DIAMETER TO BE MACHINED.
2 INSERT NUMBER TAB 1
3 MOVE LEVER DOWNWARD TO SHIFT GEARS.
4 MOVE LEVER TO LEFT TO ENGAGE FORWARD CLUTCH (TO RIGHT FOR REVERSE).
5 LEVER SPRINGS TO "UP" POSITION WHEN RELEASED, PERMITTING PRESELECTION OF NEXT SPEED, WHILE MACHINE IS WORKING.
6 PRESELECT THE NEXT SPEED BY TURNING DIAL TO CORRECT FEET PER MINUTE FOR THE SECOND DIAMETER.
7 INSERT NUMBER TAB 2
8 WHEN FIRST CUT IS COMPLETED, MOVE LEVER TO NEUTRAL POSITION . . . THEN PROCEED AGAIN AS AT 3 4 ETC.

Setting Pre-Selector for Surface Speeds.

Fig. 566

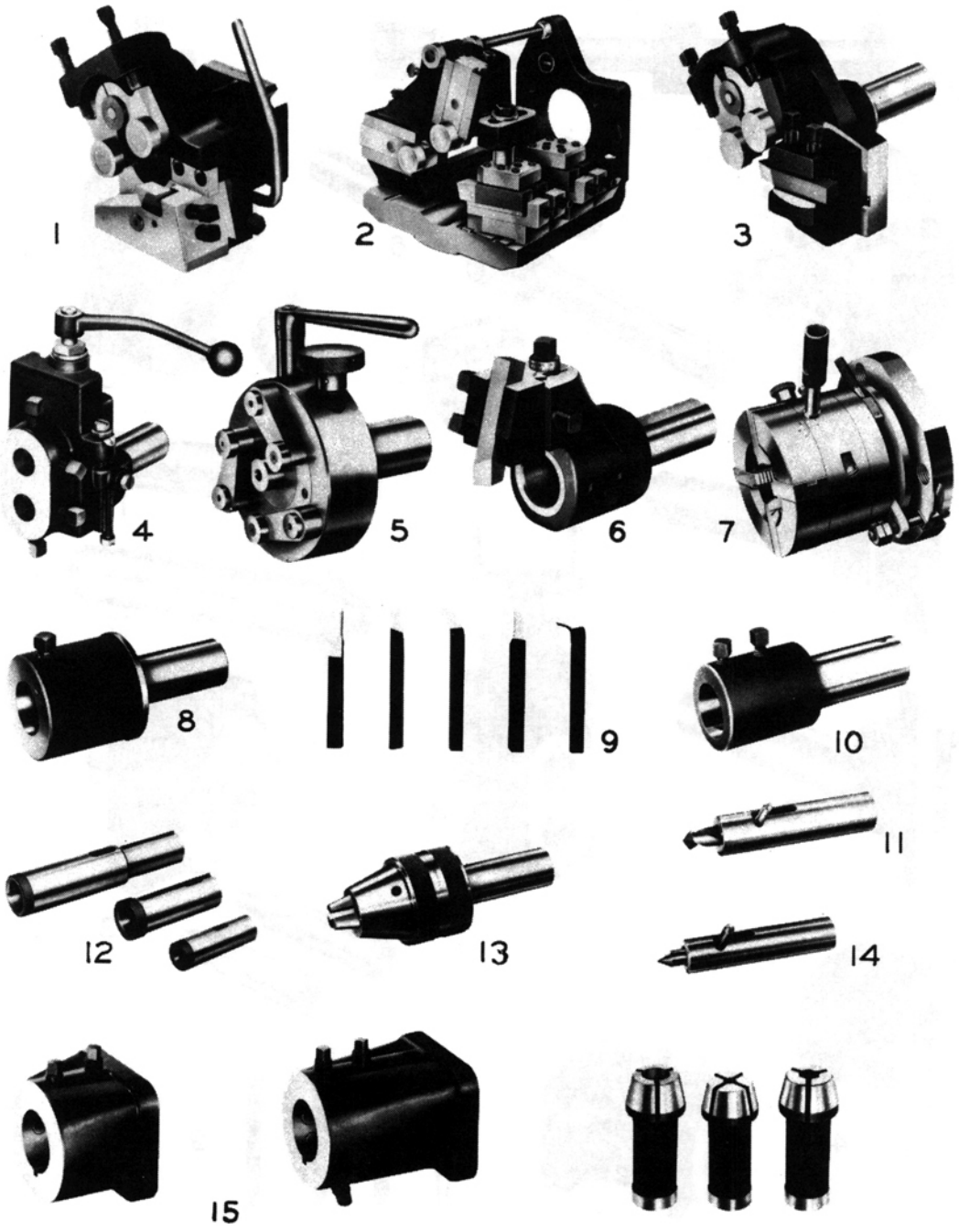


Fig. 567. Universal Tooling Equipment for Bar Work

1. Single Cutter Turner
2. Multiple Cutter Turner
3. Combination End Facer and Turner
4. Quick Acting Slide Tool
5. Center Drilling Tool
6. Adjustable Knee Tool
7. Die Head
8. Clutch Tap and Die Holder
9. Forged Cutters for Square Turret
10. Floating Tool Holder
11. Combination Stock Stop and Starting Drill
12. Taper Drill Sockets
13. Drill Chuck
14. Combination Stock Stop and Center
15. Flanged Tool Holders (Short and Long)

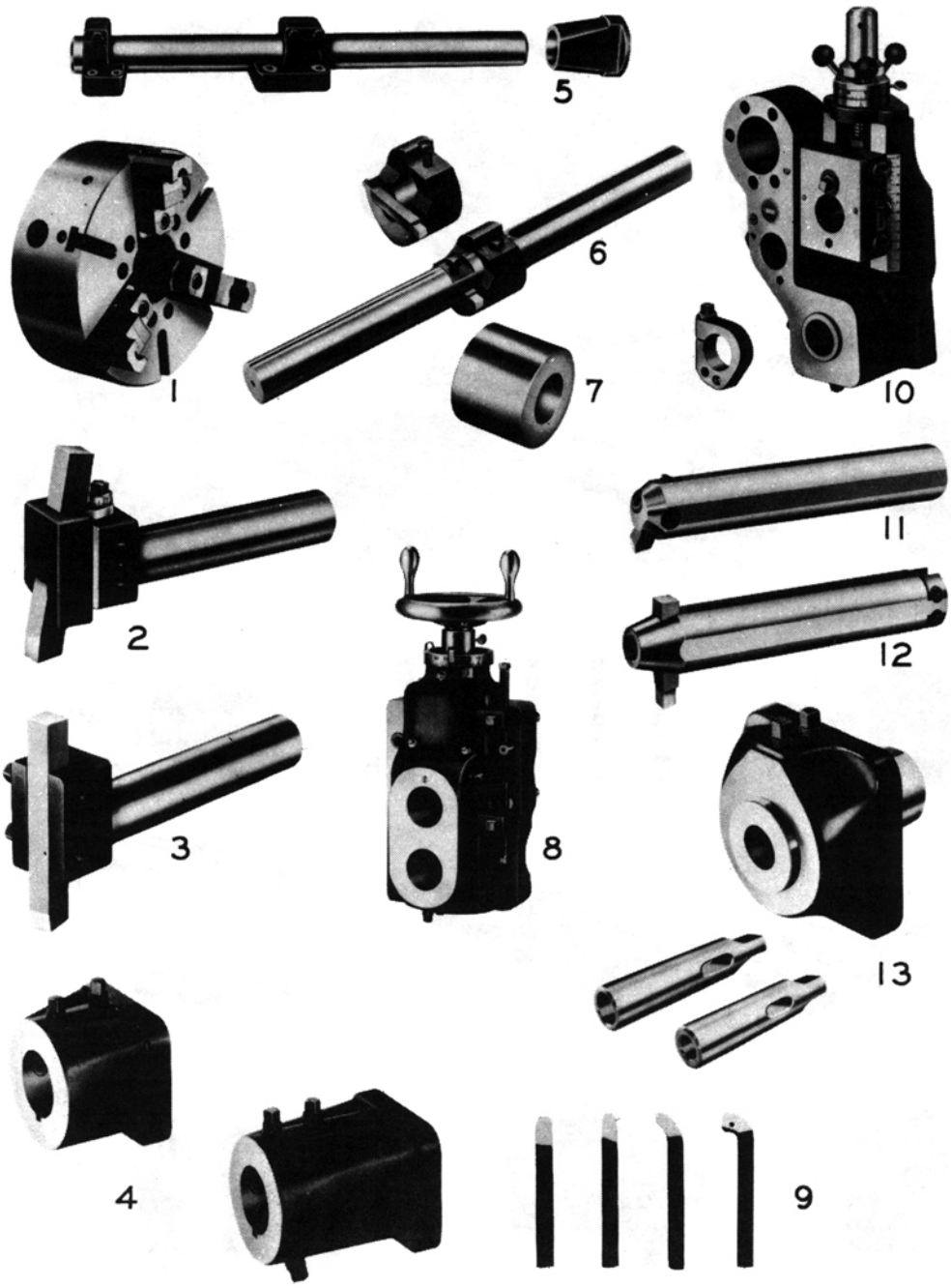


Fig. 568. Universal Chucking Tools

1. Three-Jaw Geared Scroll Chuck
2. Reversible Adjustable Angle Cutter Holder
3. Reversible Straight and Angle Cutter Holder
4. Flanged Tool Holders (Short and Long)
5. Stationary Overhead Pilot Bar and Pilot Sleeves
6. Piloted Boring Bar
7. Spindle Pilot Bushing
8. Slide Tool
9. Forged Cutters for Square Turret
10. Adjustable Single Turning Head
11. Angular Cutter Stub Boring Bar
12. Straight Cutter Stub Boring Bar (Double Ended)
13. Straight Shank Taper Socket and Holder with Taper Shell Sockets

A permanent set-up of the Universal Tooling Equipment can be arranged so that the large and heavy tools of the flanged type are permanently mounted in their logical order on the machine. If necessary for certain jobs, the tool stations can be back-indexed, or skip-indexed, to suit the requirements of the job, but the flanged tools themselves are not changed from one turret face to the other. Ordinarily, the extra machine-handling time required to skip-indexed is less than the time would be required to remove these tools from the machine or to change their position on the turret.

The lighter tools are of the shank type, which can be quickly mounted in the turret or holder.

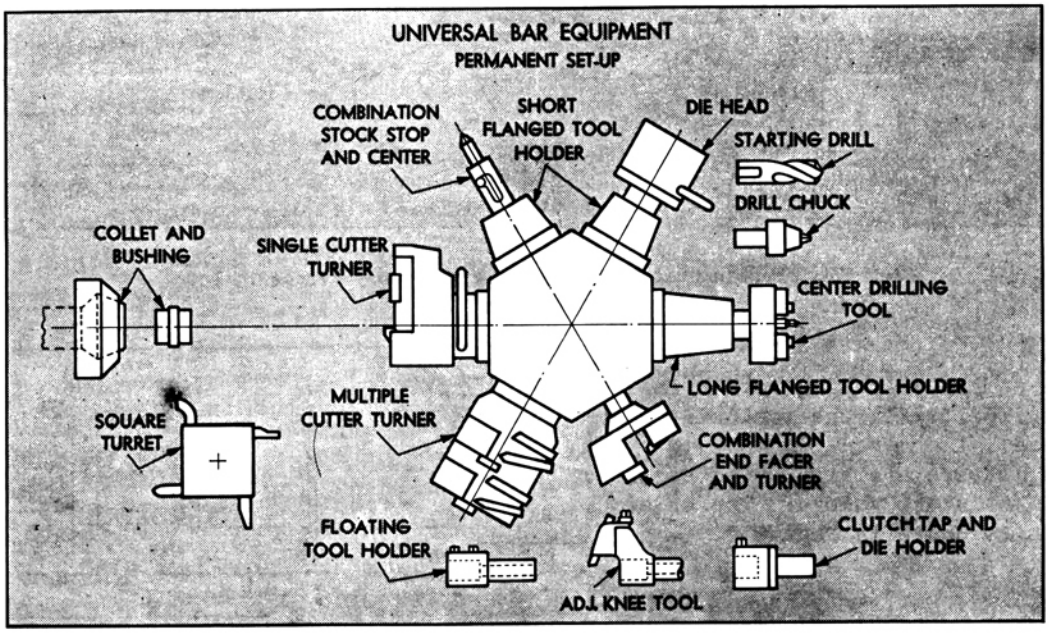


Fig. 569. Permanent Set-Up for Bar Work

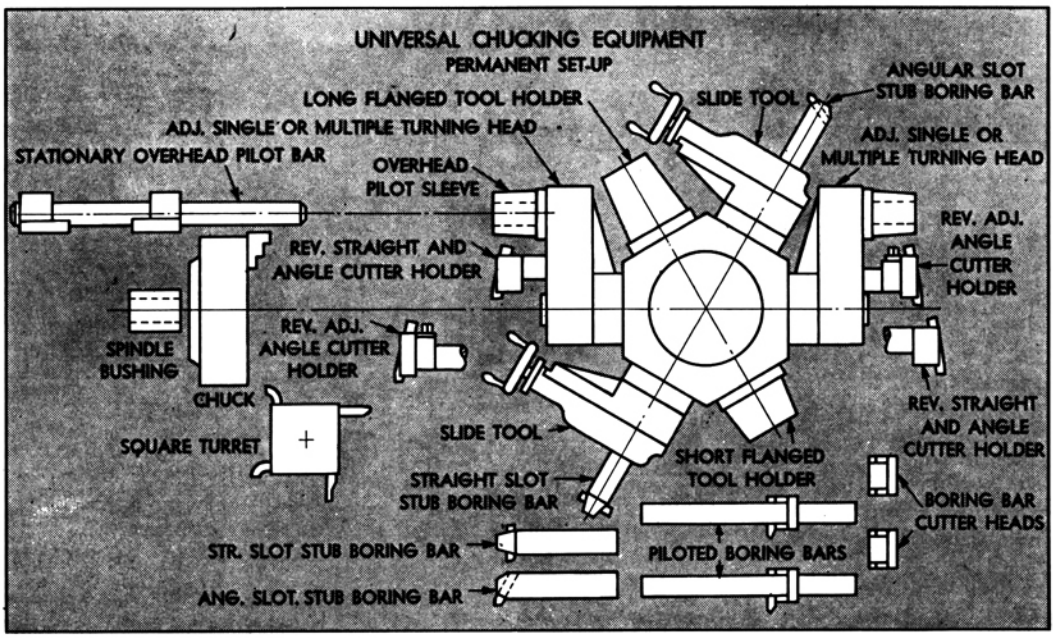


Fig. 570. Permanent Set-Up for Chucking Work

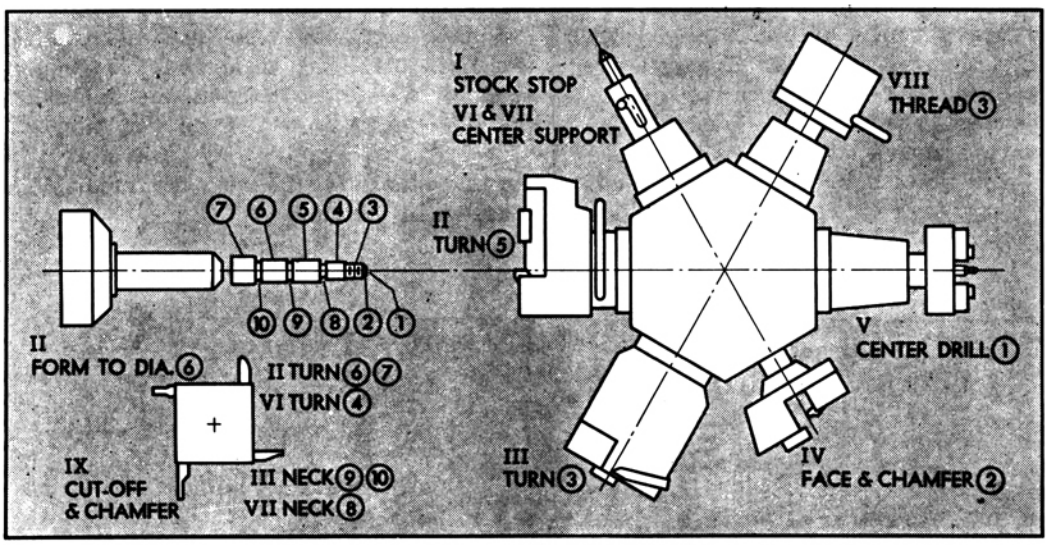


Fig. 571. Tooling Set-Up for Shaft with Twenty Pieces in Lot

The complete tooling layout is shown in Fig. 571. All the tool stations are used.

The cutters are high speed steel and of standard stock shapes. The order of operation is shown in Fig. 573.

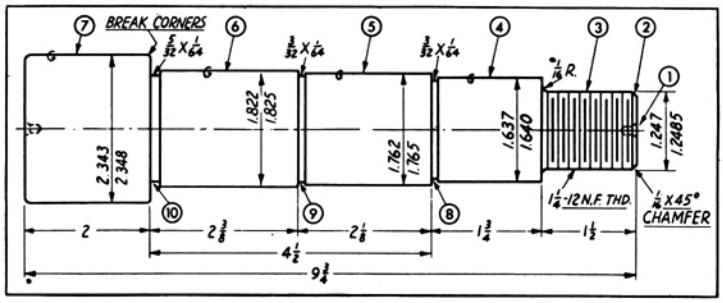
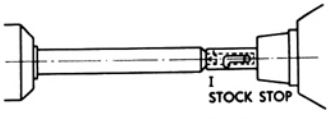
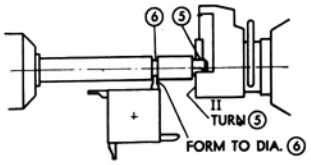
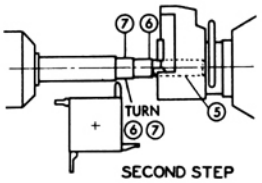


Fig. 572. Typical Bar Job--a Shaft

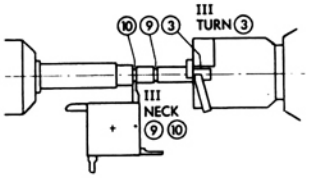


— *Operation I.*

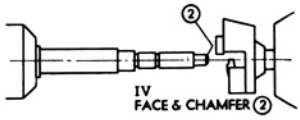




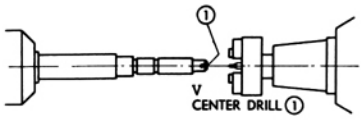
— *Operation II.*



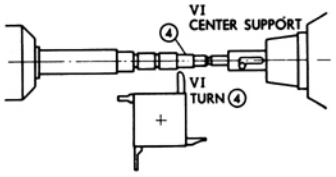
— *Operation III.*



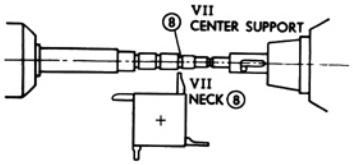
— *Operation IV.*



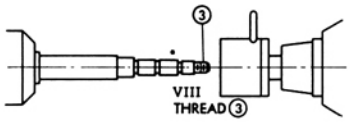
— *Operation V.*



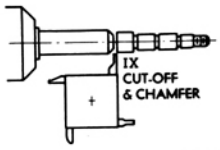
— *Operation VI.*



— *Operation VII.*



— *Operation VIII.*



— *Operation IX.*

Fig. 573. Order of Operation

THE STATIONARY OVERHEAD PILOT BAR

The Stationary Overhead Pilot Bar, Fig. 574, which fastens to the head of the machine, can be heavier than the pilot bars which are attached to the turning heads because this bar does not add any weight to the hexagon turret and its tools. This type of pilot bar can be adjusted endwise in the machine for various lengths of work, and clamped in place. The purpose of the overhead pilot bar is to tie the turret of the machine to the headstock to provide greater rigidity, and to allow deeper cuts to be taken with heavier feeds.

When the accuracy requirements of the job are studied, it is found that the diameters do not have to be turned to very close limits since they are to be ground after hardening. However, it is necessary to keep the diameters closely concentric in order to allow for "clean-up" in the grinding operations. The thread diameter, which is not ground, must run true with the other diameters.

VERTICAL TURRET LATHE

The vertical lathe in Fig. 577 is a machine having a horizontal rotary table with adjustable chuck jaws and a vertical turret. It is adapted for turning, boring, facing, drilling, reaming, and similar operations. A thread cutting attachment is also used on some of these machines.

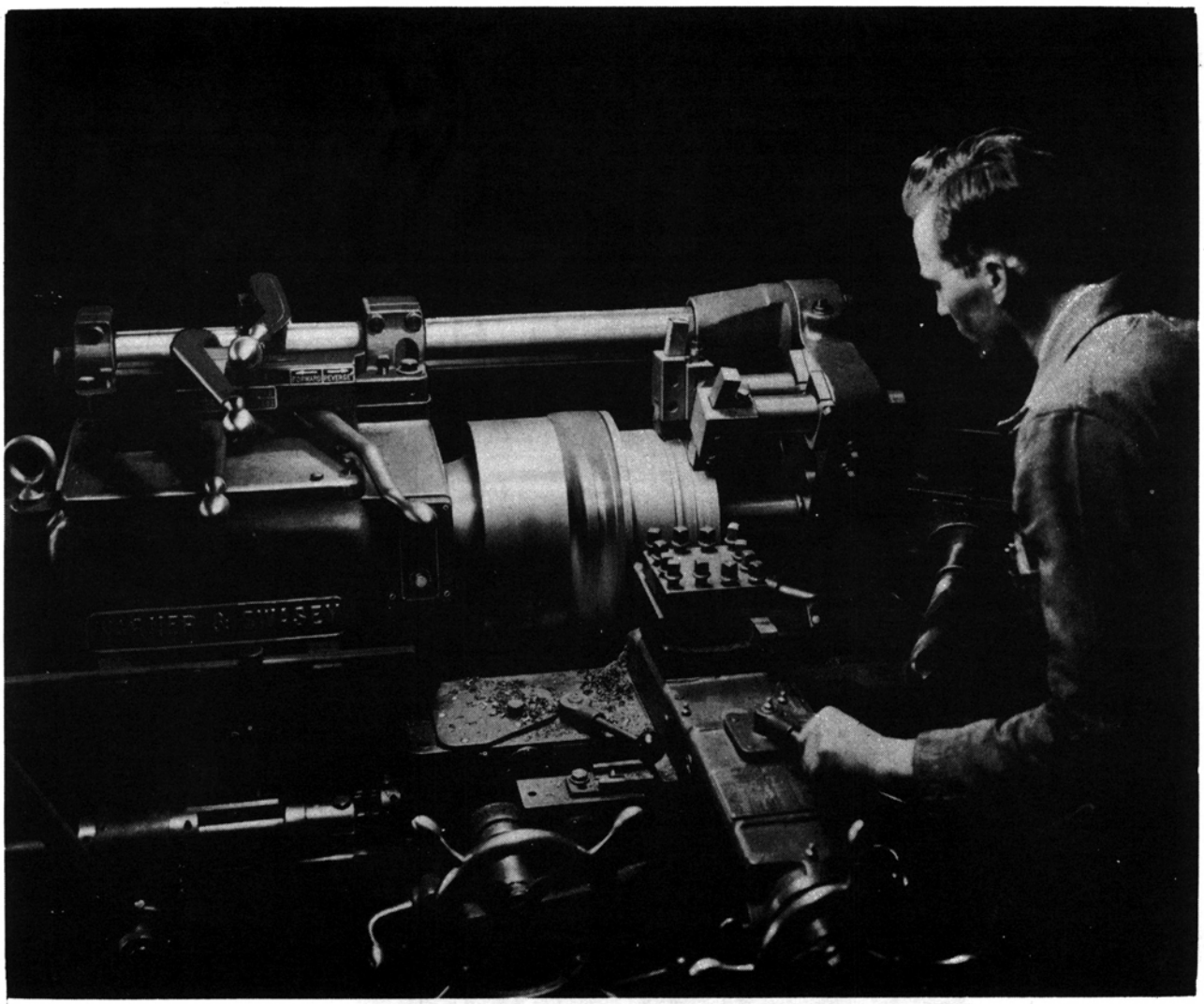


Fig. 574. Combined and Multiple Cuts

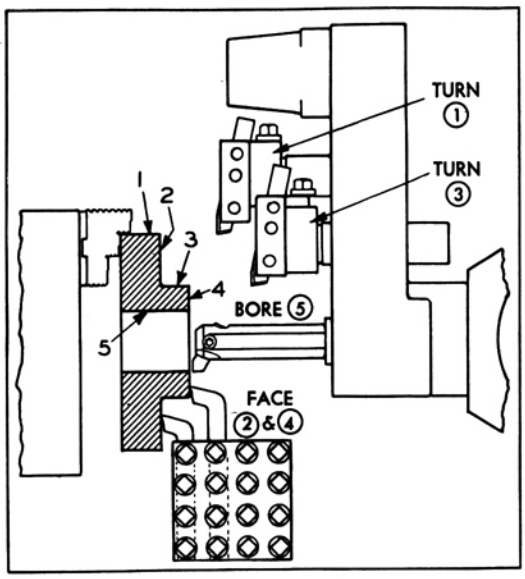


Fig. 575. Combined and Multiple Cuts Being Taken on a Simple Gear Blank Job in Quantity Lots.

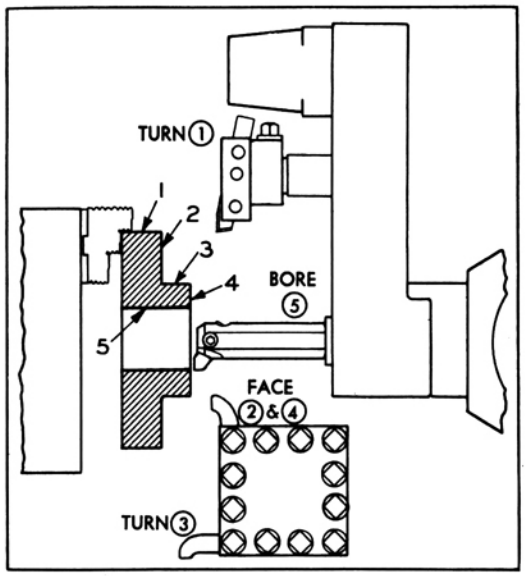


Fig. 576. Combined and Multiple Cuts Being Taken on a Simple Gear Blank Job in Small Lots.

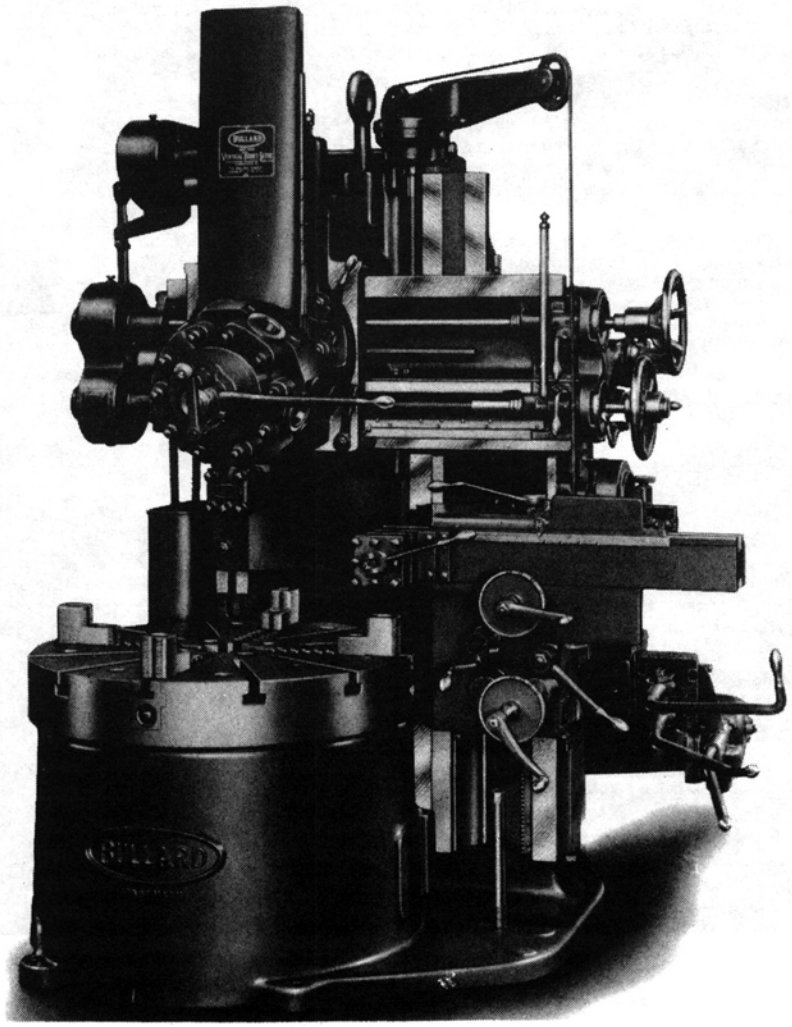


Fig. 577. Vertical Turret Lathe

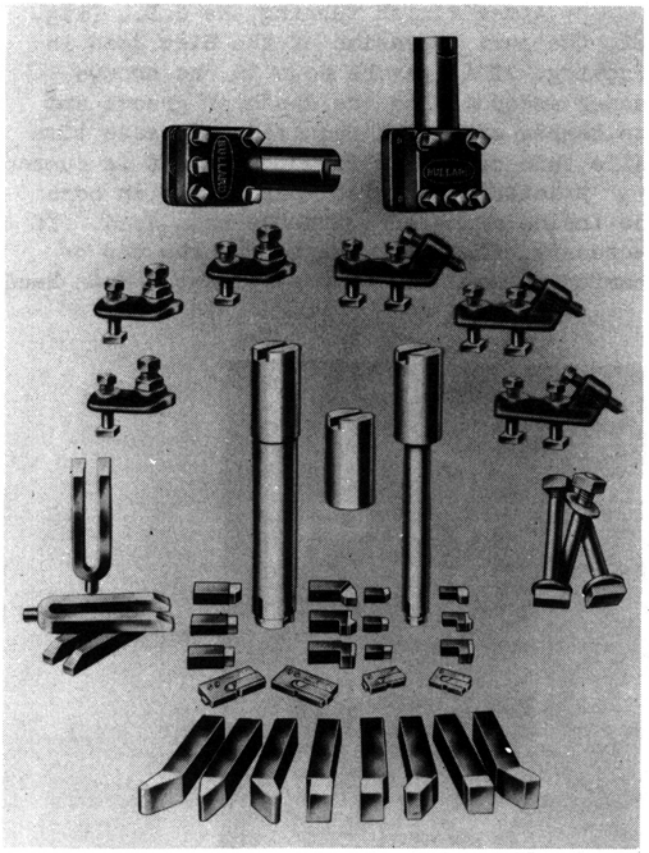


Fig. 578
Standard Tool Equipment for Vertical Turret Lathe

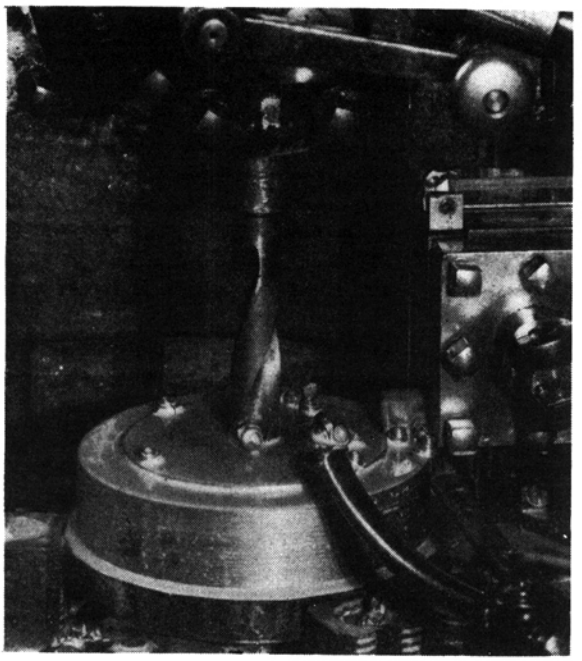


Fig. 580
Drilling and Machining Operation

The illustration (Fig. 580) shows machining from solid stock. Main Head operations are drilling, rough bore, finish bore, first and second ream with a chamfer or burring operation. Simultaneously the Side Head takes a rough and finish face, chamfer, rough and finish turn. The sequence of operations and the simultaneous use of Side and Main Heads provide a method for cutting waste time between cuts.

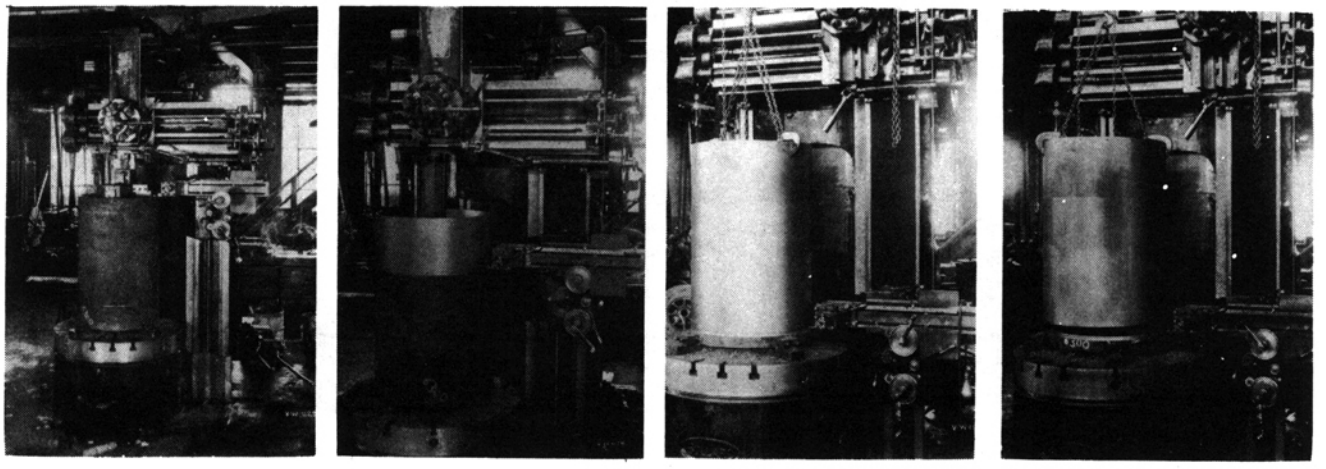


Fig. 579. Machining a Cylinder on the Vertical Lathe

In as much as the angles shown in Fig. 581 require accuracy and fine finish, a rough cut is first taken with the Side Head Ram, using a single point cutter. To do this, the Main Head is indexed to present the cam located in a holder of the turret. The Side Head then feeds against this cam forcing the special tool holder downward at the cam angle, thus taking a rough angular facing cut. To obtain the fine finish and accuracy, the Main Head is then indexed to present the sweep tools for the two angles.

After finish turning the O.D., (Fig. 582, the next operation of the Side Head is grooving. This may be done in one or two passes according to the depth of groove and the degree of finish required. To save time while this operation is going on, it is customary practice to finish turn or finish bore the inside diameters with the Main Head. If necessary, finish facing across the top or chamfering may also be done with the Main Head.

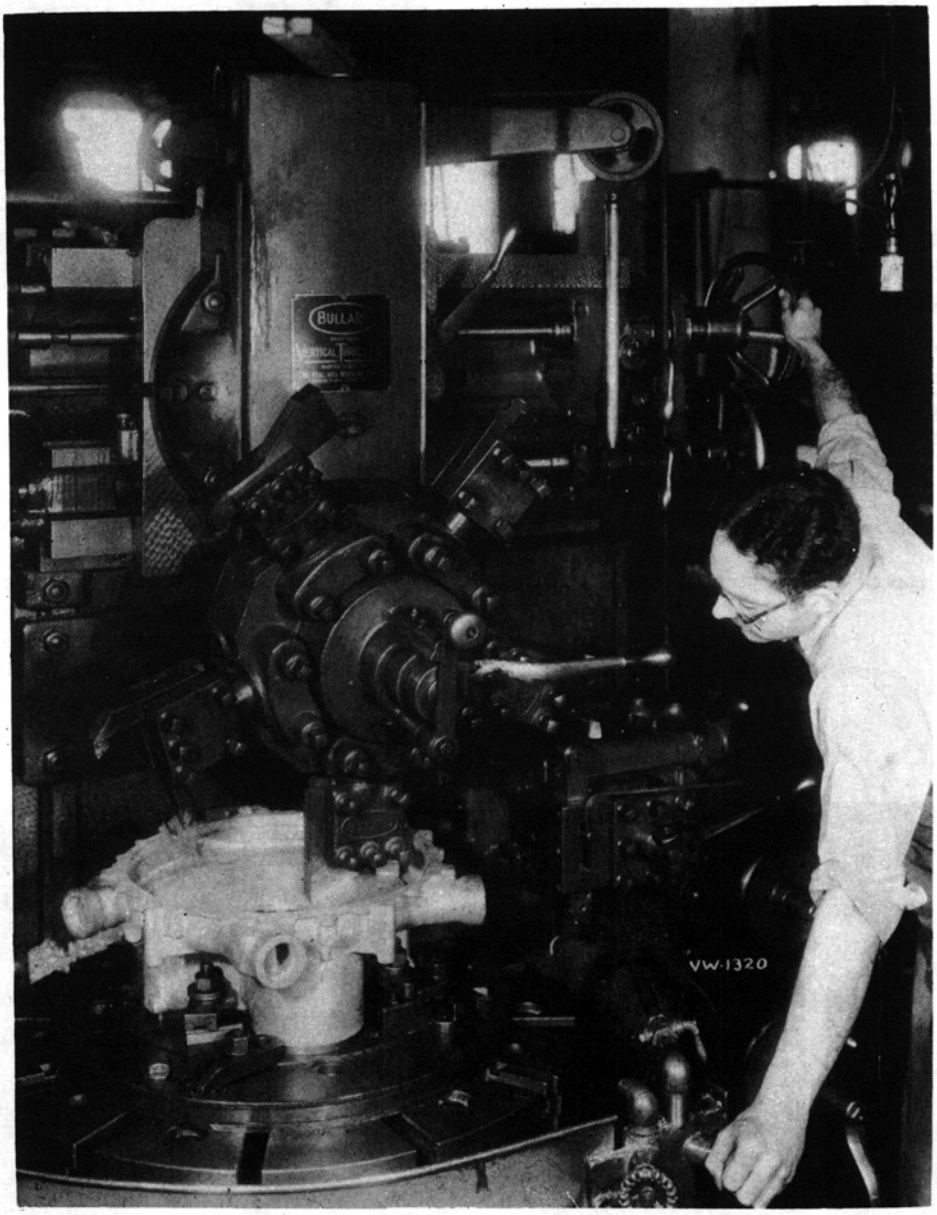


Fig. 581
Angular Facing Operation

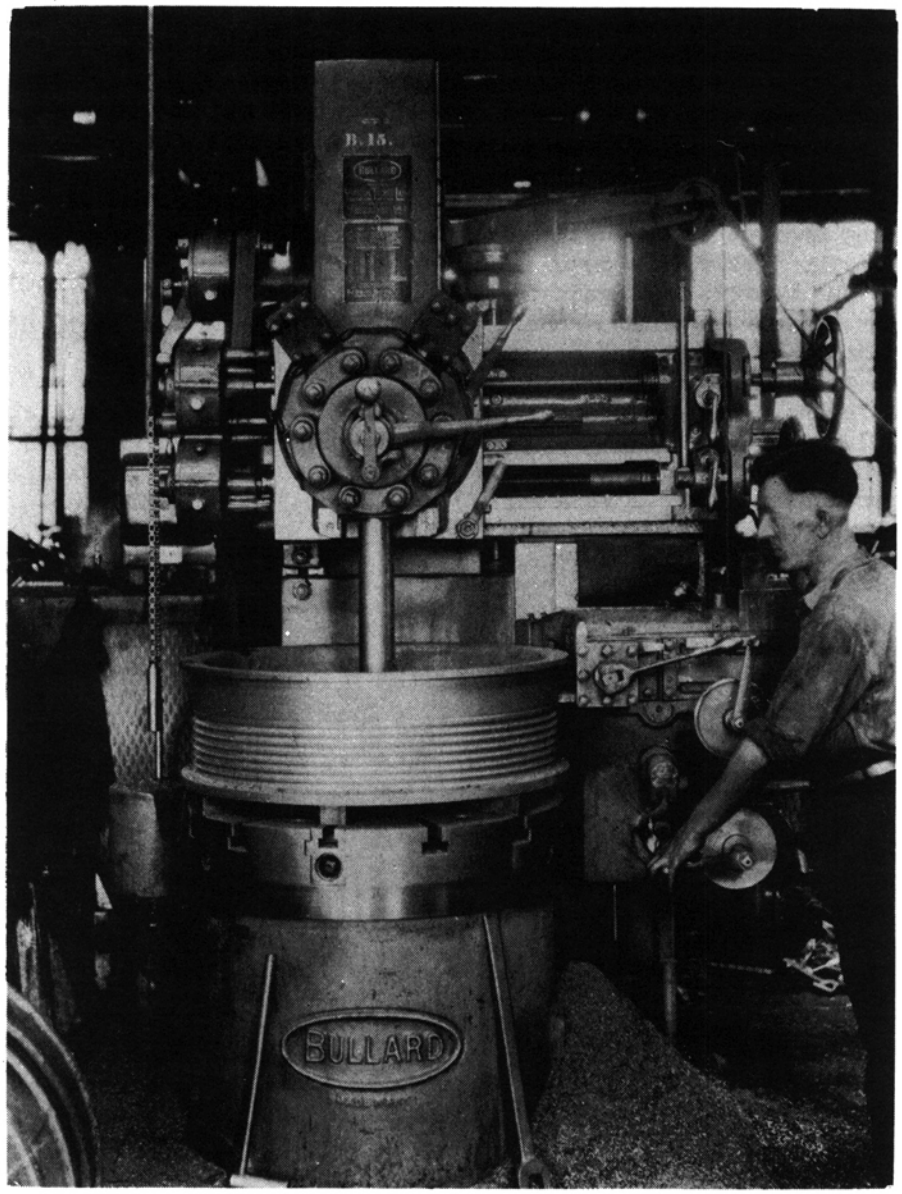


Fig. 582
Grooving Operation

Chapter 21

MILLING MACHINES

The milling machine is a machine that removes metal from the work with a revolving milling cutter as the work is fed against it. The milling cutter is mounted on an arbor where it is held in place by spacers or bushings. It is made from high speed steel and can be had in different sizes and shapes. Regular or irregular shaped work may be produced on this machine, designs varying according to the particular class of work wanted. Milling machines may be grouped into various classes according to this variation in general appearance design, as the column and knee type, the manufacturing types, and the planer type.

The important parts of the milling machine (see Fig. 583) are:

1. Starting levers
2. Spindle
3. Column
4. Knee
5. Table
6. Elevating screw
7. Dividing head
8. Speed levers
9. Feed levers
10. Table movement levers
11. Foot stock
12. Arbor yoke

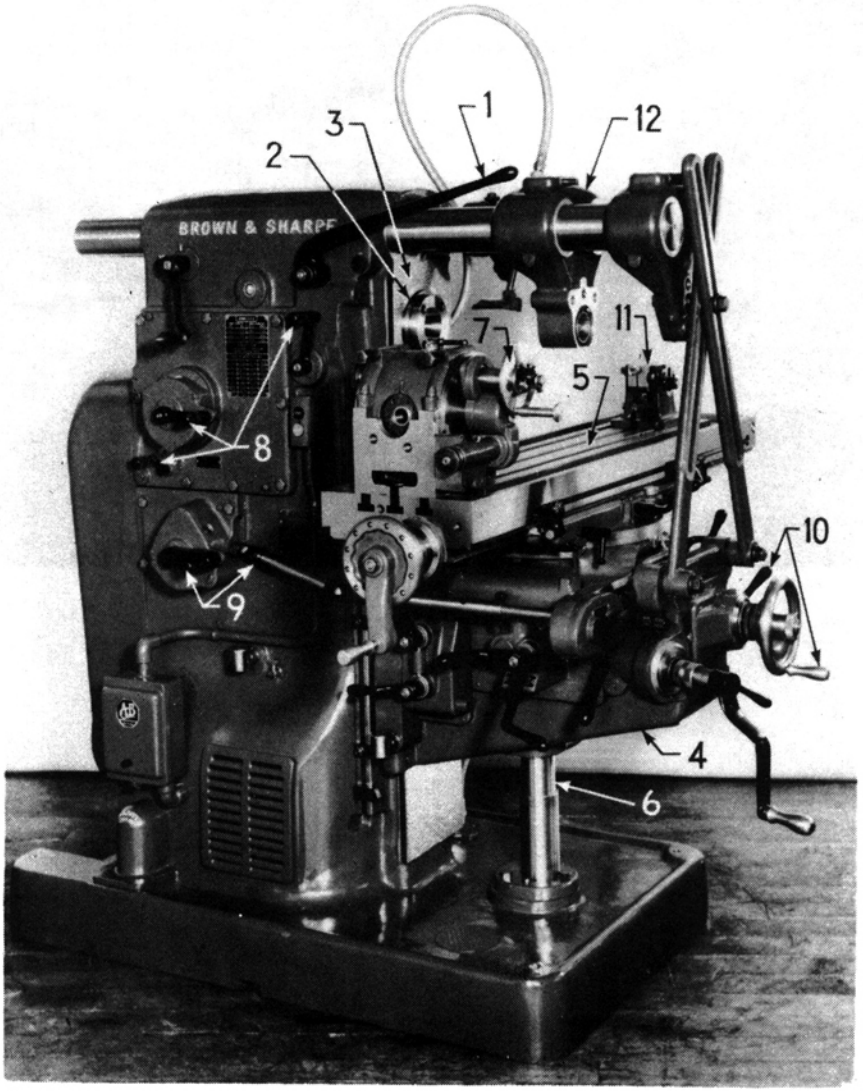


Fig. 583. Universal Milling Machine--Column and Knee Type

The most common type of milling machine used in the tool room is the "column and knee" type. This machine is so-named because of its design. The main casting consists of a high column, to which is fastened a bracket or "knee," which supports the table. The knee is adjustable on the column, so that the table can be raised or lowered to accommodate jobs of various sizes. Vertical cuts may be taken by feeding the table up or down. The table may be moved in the horizontal plane in two directions, either at right angles to the axis of the spindle (longitudinal feed) or parallel to the axis of the spindle (transverse feed).

The Universal milling machine (Fig. 583) is a development that embodies all the principal features of the other types of milling machines. It is designed to handle practically all classes of milling machine work. The table has the same movements as the plain milling machine, but in addition it can be swiveled on the saddle so that it moves at an angle to the spindle in the horizontal plane. When fitted with an attachment known as an index head, angular, spiral, and helical cuts may be made. The universal type of milling machine is used to cut helical gears, twist drills, milling cutters, and various kinds of straight and taper work. The universal milling machine is regarded by many as one of the most important machines in the tool room.

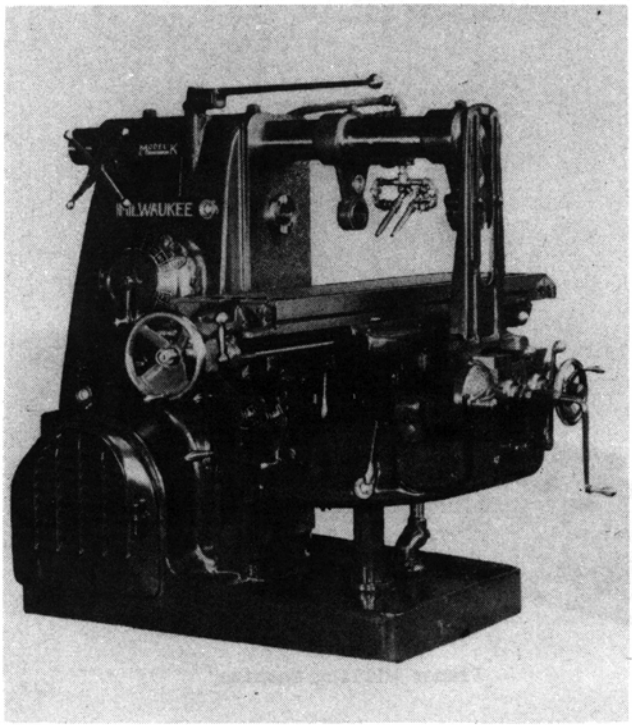


Fig. 584
Plain Milling Machine

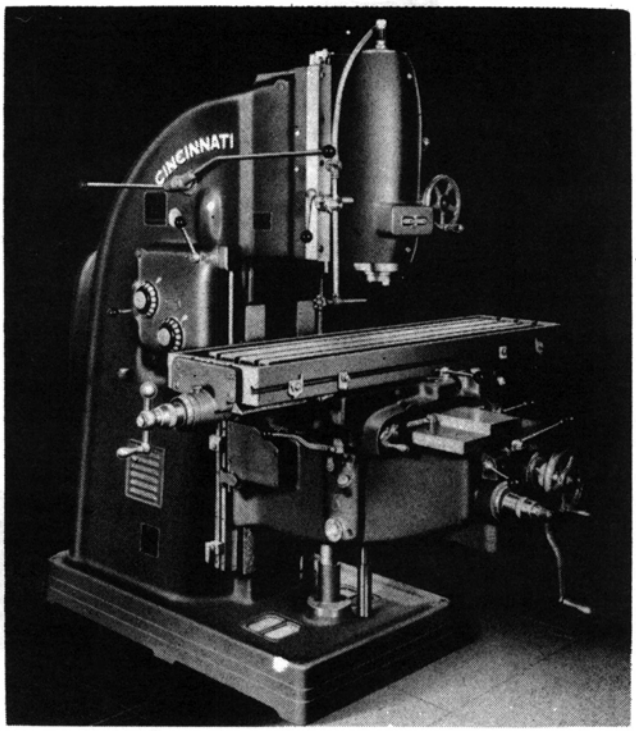


Fig. 585
Vertical Spindle Milling Machine

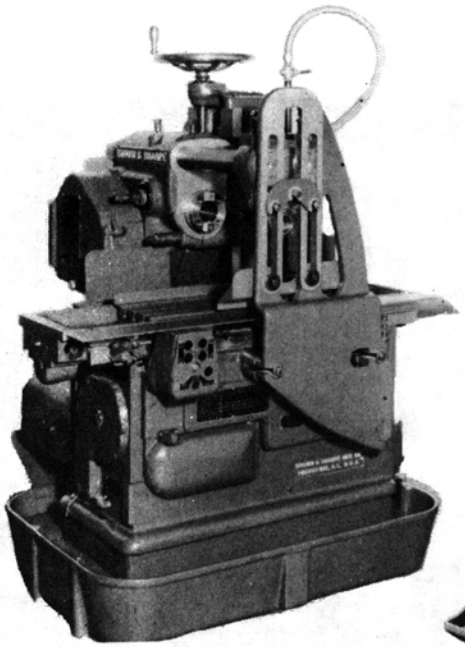
The Plain milling machine shown in Fig. 584 is one in which the longitudinal travel of the table is fixed at right angles to the spindle. In this machine the table has three movements: longitudinal (at right angles to the spindle), transverse (parallel to the spindle), and vertical (up and down). It is the practice to take heavy cuts at fast speeds and coarse feeds in the classes of work for which the medium and larger sizes of plain milling machines are adapted. The rigid construction of the machine enables this to be done successfully, and it is this ability that is the chief value of the plain machine.

The Vertical Spindle milling machine shown in Fig. 585 embodies the principles of the drilling machine. The spindle and table are similarly located, and the cutter is mounted in the spindle. The spindle has a vertical movement, and the table has vertical, longitudinal, and transverse movements. The vertical spindle milling machine is used for face milling, profiling, die sinking, and for various odd-shaped jobs. Owing to the position of the spindle, this type of machine can be used advantageously in boring holes.

The Manufacturing milling machine shown in Fig. 586 is used primarily for the production of small parts of typewriters, sewing machines, and similar machine and tool parts. The spindle is supported in bearings located in the adjustable head, which can be raised or lowered. The capacity of the machine is limited for work of great height because the table cannot be raised or lowered, the only adjustment being in the head. Furthermore, there is no transverse table feed, the only transverse movement being obtained by a slight adjustment of the spindle.

The Planer milling machine shown in Fig. 587 is designed for the heaviest classes of slab and gang milling. It bears a marked resemblance to the planer, from which it derives its name. The spindle is mounted in bearings carried in a vertically adjustable slide similar to that of a planer, and is in a corresponding position. The class of work produced on the planer mill is identical to that of the column and knee type machine, therefore the same principles are involved.

The "Precision" Horizontal Boring, Drilling and Milling Machine, Fig. 588, is used for performing any or all of the stated operations on a piece of work at a single



Manufacturing Milling Machine

Fig. 586

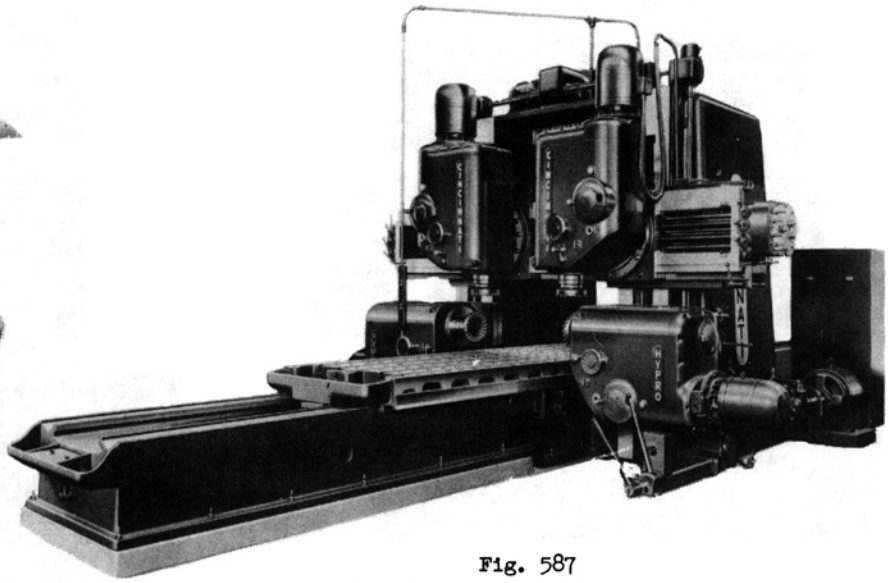


Fig. 587
Planer Milling Machine

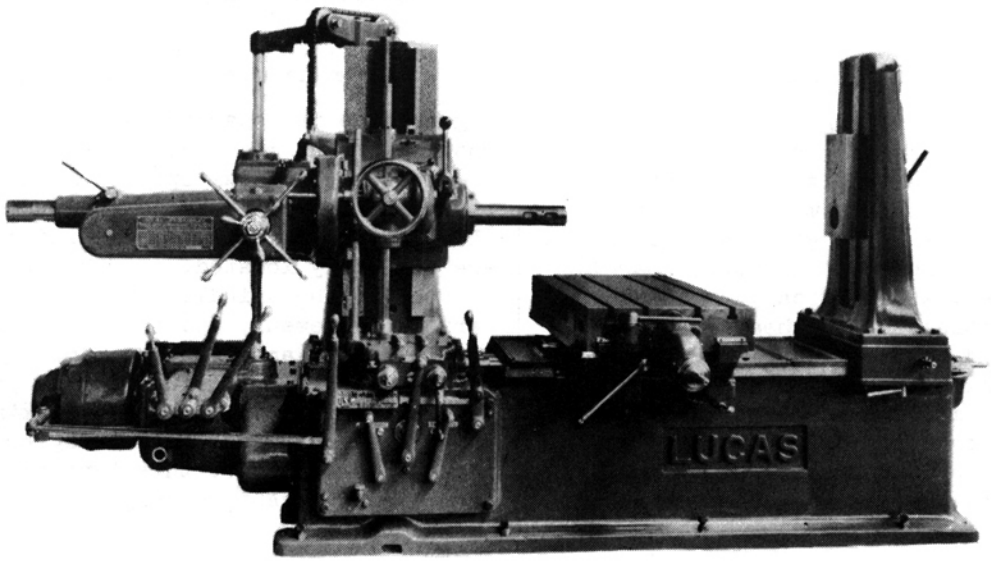


Fig. 588. Horizontal Boring Machine

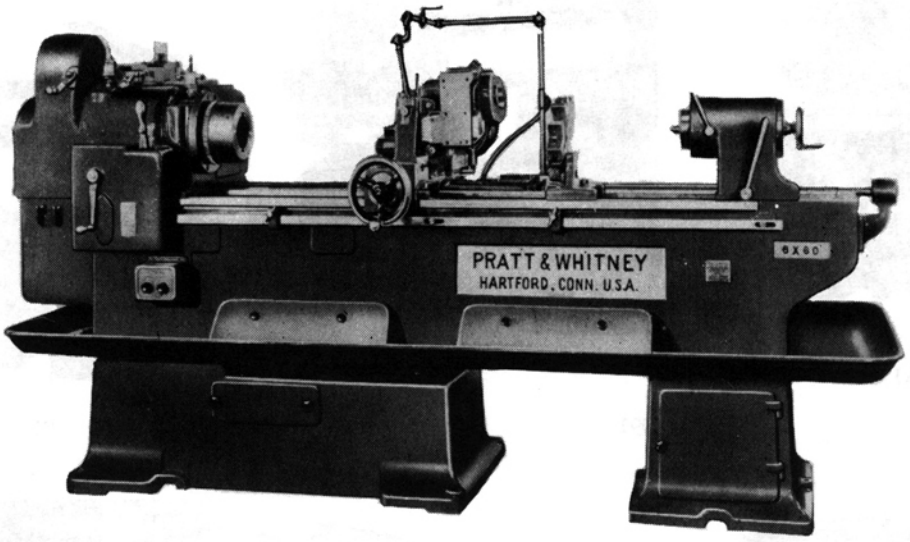


Fig. 589. Thread Miller

setting. This saves time in resetting the work and insures accuracy in relation of the machined surfaces to each other. The Thread Mill, Fig. 589, machines the different kinds of standard threads, using a special cutter.

Fig. 590 shows the Spline Mill used to mill slots and keyways with either open or closed ends.

In order to increase the range of work that can be done on a milling machine, many mechanisms known as attachments are used. Some of the most commonly used attachments are discussed in the following paragraphs.

The Compound Vertical Milling Attachment, Figs. 591(a) and 591(b), is particularly applicable to a large variety of milling, because it can be set in two planes. It is especially advantageous when it is desired to set the spindle at an angle to the table, as in milling angular strips, table ways, etc., for with the spindle in this position, the full length of the table travel is available, and an ordinary end mill, instead of an angular cutter, can be used for milling the angle.

The Universal Milling Attachment, Fig. 592, as its name implies, is fully universal in regard to setting the spindle. Its range of work is very much the same as is covered by the preceding attachments, and, in addition, it can do many unusual jobs because its spindle may be set at any angle in both planes. The outer end of this attachment is so made that an arbor yoke can be used to provide additional stability.

Rotary Attachments are used on a variety of circular milling such as circular T-slots, segment outlines, etc., and on a great deal of tool and die-making jobs that require splining, slotting, or irregular form milling. Fig. 593 shows a hand feed unit, and Fig. 594 a unit equipped with a power feed. Both attachments are graduated on the circumferences of the tables, and both have adjustable dials on the worm shaft.

The Slotting Attachment, Fig. 595, is

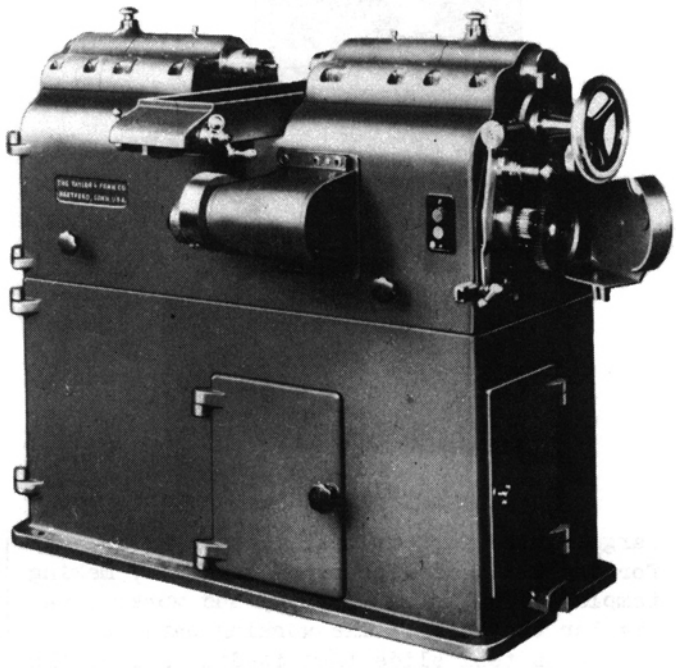
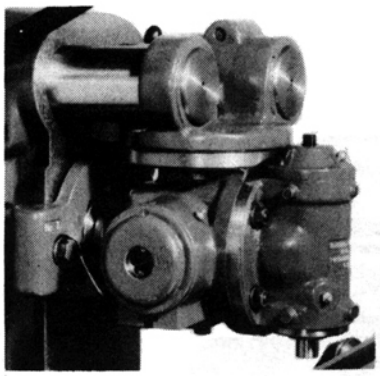
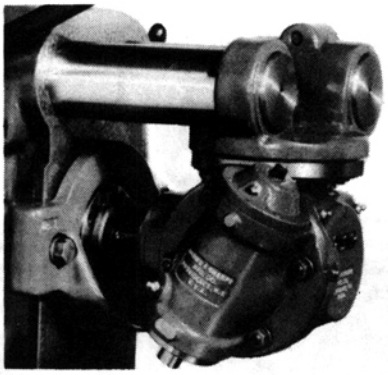


Fig. 590. Spline Mill



(a)



(b)

Fig. 591

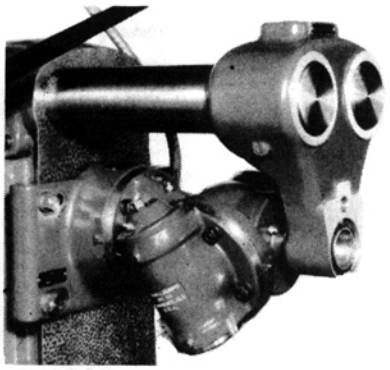


Fig. 592

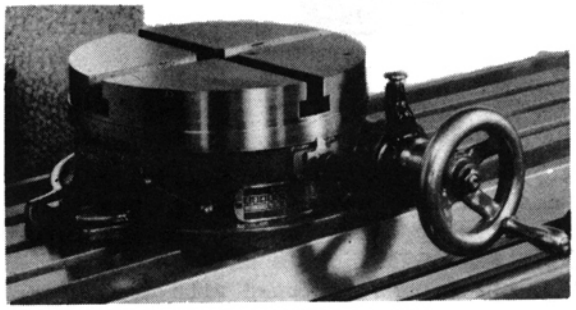


Fig. 593

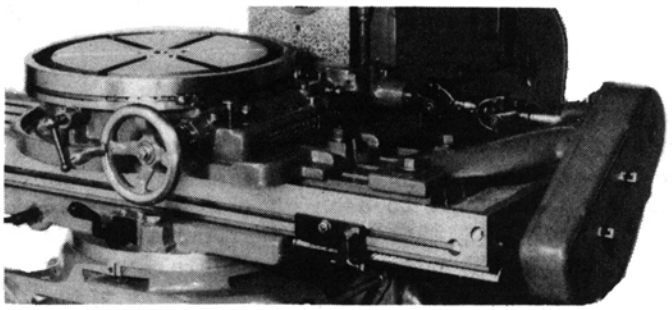


Fig. 594

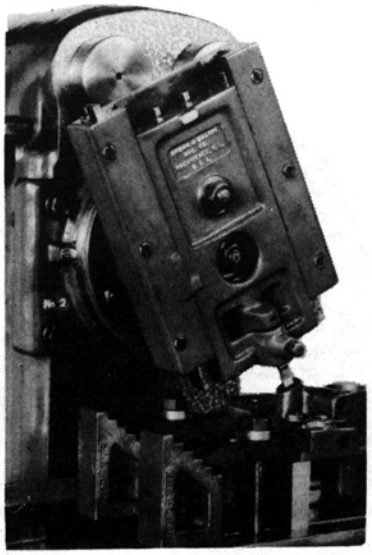


Fig. 595

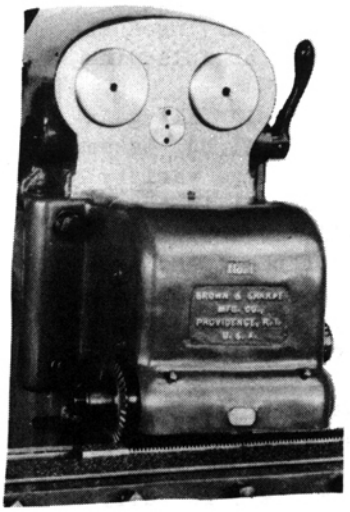


Fig. 596

largely used in tool making, such as in forming box tools for screw machines, making templates, splining keyways, and work of a similar character. The working parts consist of a tool slide that is driven from the machine spindle by an adjustable crank that allows the stroke to be set for different lengths. The attachment can be set at any angle between 0 and 90°, either side of the center line, the position being indicated by graduations on the circumference of the head. The tool is held in place by a clamp bolt, and a stop that swings over the top of tool shank making it impossible for the tool to be pushed up.

The Rack Milling Attachment, Fig. 596, is used for cutting teeth in racks. It can also be used in connection with the Universal Spiral Index Centers for cutting worms, on Universal Milling Machines, and for other miscellaneous operations. The cutter is mounted on the end of a spindle that extends through the attachment case parallel to the table T-slots. This

spindle is driven from the machine spindle by a train of hardened steel bevel and spur gears.

The High Speed Milling Attachment, Fig. 597, consists of a pair of gears for increasing the speed, and an auxiliary spindle that drives the cutter. It is used in order to obtain the correct speed for small milling cutters which should be run more rapidly than the fastest spindle speed when cutting keyways and slots, etc.

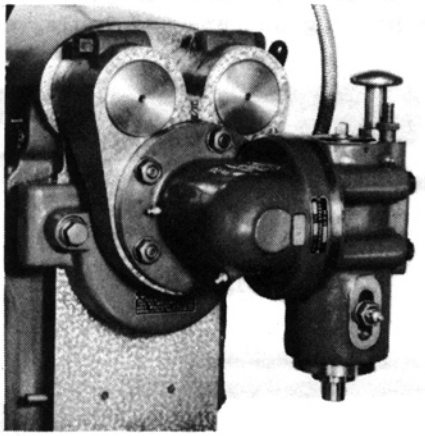


Fig. 597

The Tilting Table, Fig. 598, is designed primarily for use in connection with index centers when fluting taper reamers, taps, etc. In addition to this work, many other kinds of taper pieces can be accurately reproduced.

Fig. 598

Micrometer spindle and micrometer table setting attachments are used to facilitate the performing of extremely accurate boring and milling jobs. They are of particular advantage in making jigs, fixtures, dies, and other tool room work where accurate adjustments are required. Measuring bars used in connection with micrometer heads and dial gages are employed. These bars may be used in combinations to obtain various adjustments.

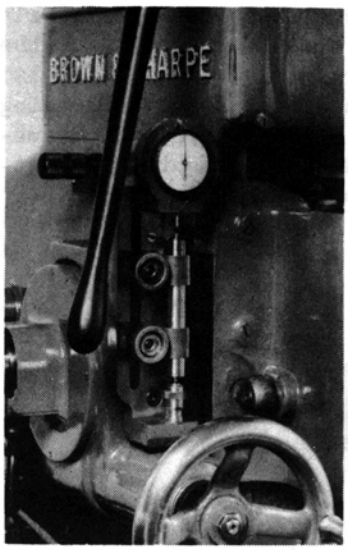


Fig. 599

Fig. 599 shows a micrometer spindle setting attachment.

The Index Head, Fig. 600, is used for obtaining equally spaced divisions on the periphery of work such as gears, drills, reamers, cutters, etc., and also for helical, spiral, and general taper work.

Triple Index Centers, Fig. 601, are used for manufacturing purposes where economy

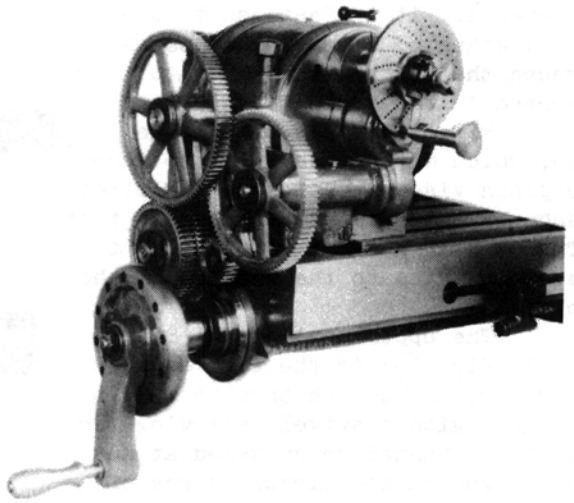


Fig. 600

Fig. 601

and rapid production are important factors. All the spindles are operated simultaneously by moving the index crank. The ratio of the worm to the worm-wheel is 40 to 1, which is the same as the usual index head. When using the three spindles, a job 4 inches in diameter can be swung, and when using the two outside spindles, a job 8 inches in diameter can be swung.

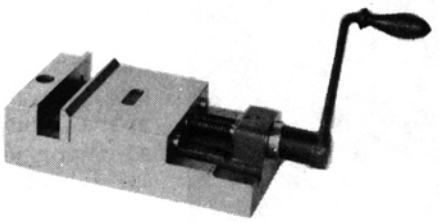


Fig. 602

The Plain Vise, Fig. 602, is used for light milling operations. The bed and slides are made of cast iron, and the jaws are made of tool steel, hardened and ground. It is fastened to the table by means of a screw that passes through the bed and threads into a nut inserted in the table T-slot.

Fig. 603 shows the Flanged Vise, which differs very little from the plain vise, except that a slotted flange is provided at each end so that a T-slot bolt and nut can be used to fasten the vise to the surface of the table.

The upper part of the Swivel Vise in Fig. 604 is the same as the flanged vise, but the base is held to the table with a swivel, allowing the vise to be turned or swiveled at any angle. The entire circumference of the base is graduated in degrees.

The Toolmakers' Universal Vise in Fig. 605 is of great advantage to the toolmaker, as a job can be set at any angle in the horizontal or vertical plane.

The Spring Chucks for milling machines shown in Fig. 606 are convenient for holding wire, small rods, straight shank drills, mills, etc. The collet holder is made of steel, is ground to fit a standard tapered hole, and has a hole through its entire length. The spring collet is held in place by a cap nut that forces it against the tapered seat and closes the chuck concentrically.

The Collets shown in Fig. 607 are used for holding cutters and tools, depending

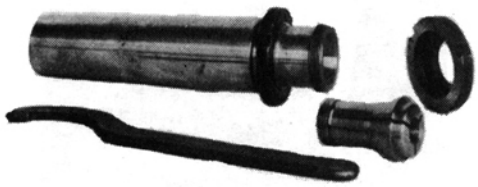


Fig. 606

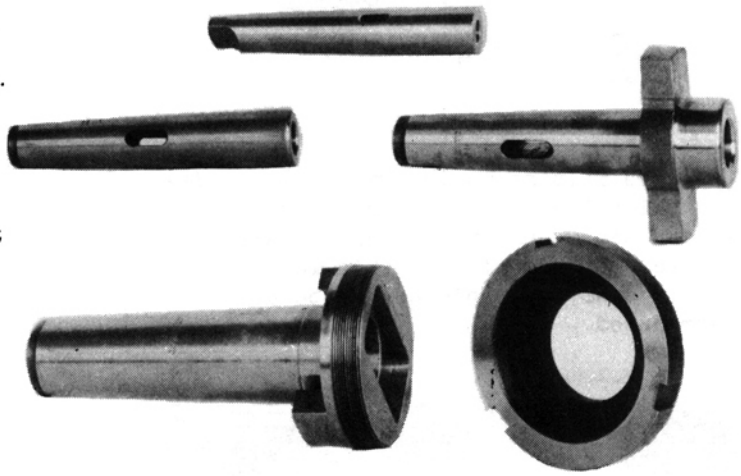


Fig. 607

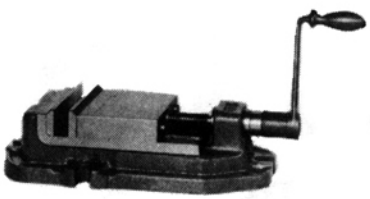


Fig. 603

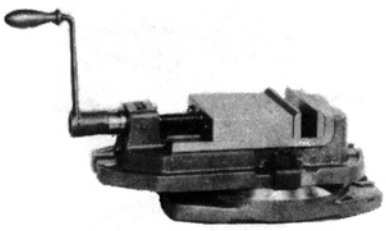


Fig. 604

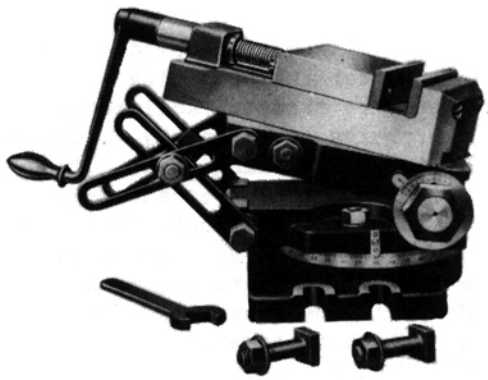


Fig. 605

upon the nature of the work. Notice that one has a tang, one has a tapped hole to receive a draw bar, one has a tenon to help in driving, and one is used in connection with a split collet and has a cap which is used to close it to hold the cutter.

Fig. 608. Old Style Spindle Arbor

Fig. 609. Standardized Spindle Arbor

Milling machine spindles were formerly made to accommodate arbors of the type shown in Fig. 608. Since 1927, when the National Machine Tool Builders' Association adopted a standard taper spindle end having a taper of $3\frac{1}{2}$" per foot, arbors like the one shown in Fig. 609 have been used.

A milling machine operator should be familiar with the different types of milling machine cutters. Figures 610 to 625, on the following page, show some of the more common types of cutters.

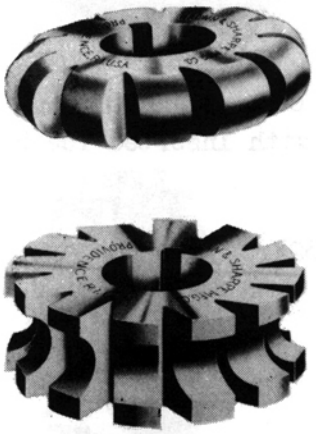


Fig. 626. (Top) Convex and (Bottom) Concave Cutters

Some end-mills are right-hand (RH) and some are left-hand (LH). To distinguish between them, hold the shank of the cutter in your hand, with the shank end toward you. If it cuts when it revolves to the right (clockwise), it is a right-hand cutter, and if it cuts when it revolves to the left (counterclockwise), it is a left-hand cutter. In helical end-mills, right-hand cutters usually have left-hand helices and left-hand cutters have right-hand helices, so that there will not be a tendency for a cutter to pull out of the spindle when cutting.

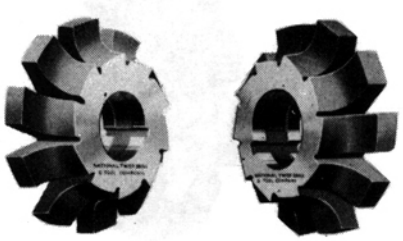


Fig. 627. Corner Rounding Cutters

Fig. 628. Straddle Tooth Sprocket Cutter

Fig. 629. Woodruff Key Seat Cutter

OPERATIONS

Fig. 630 illustrates a method of milling square or hexagon heads on bolts, using a

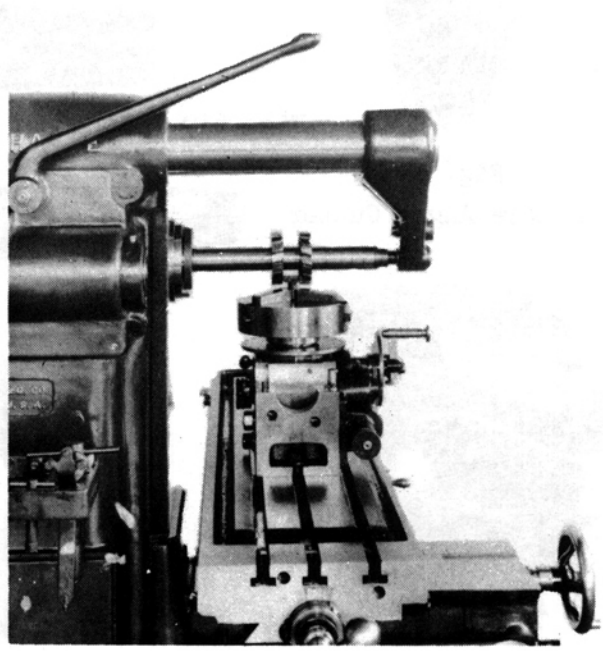


Fig. 630

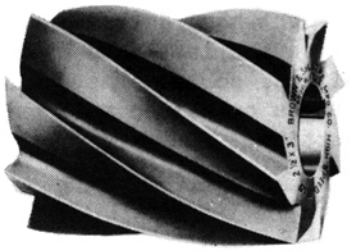


Fig. 610.
Coarse-Tooth Milling Cutter

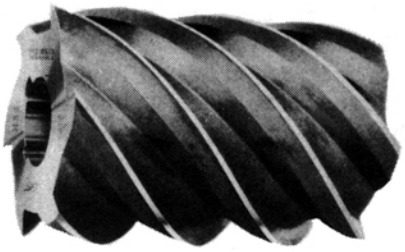


Fig. 614.
Helical Plain Milling Cutter

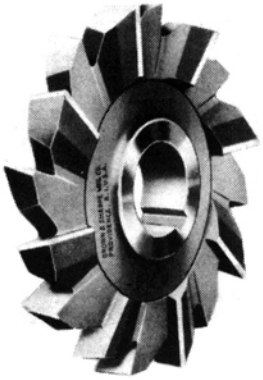


Fig. 622.
Staggered Tooth Side Milling Cutter

Fig. 615. Taper Shank Cutter

Fig. 616. Double-End Cutter

Fig. 617. Two Lipped Taper Shank

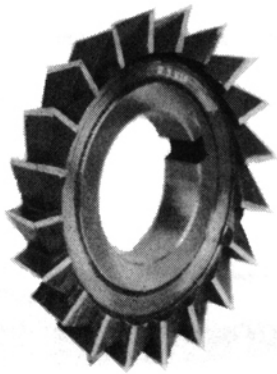


Fig. 611.
Angular Cutter

Fig. 618. Milling Machine
Standard Taper Shank

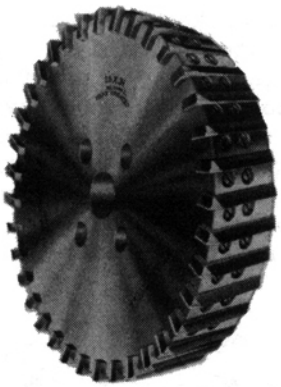


Fig. 623. Face Milling Cutter with Inserted Teeth

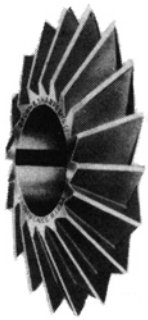
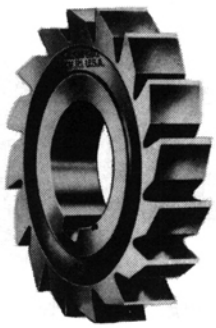


Fig. 612.
Double Angle Cutter

Fig. 619. Straight Shank Spiral End Mill

Fig. 624.



Plain Milling Cutter

Fig. 620. T-Slot Cutter

Fig. 613. Shell End Mill

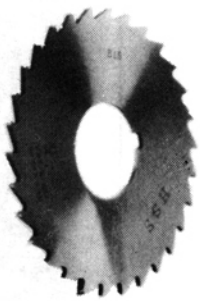


Fig. 621.
Metal Slitting Saw

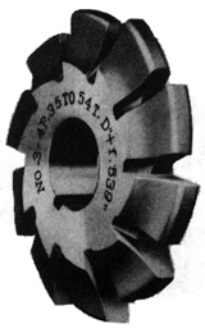


Fig. 625.
Involute Spur Gear Cutter

chuck on the dividing head spindle for holding the work. This figure also furnishes a good example of the use of a pair of side milling cutters as "straddle mills." Two sides are finished at a cut, completing a square bolt head with two cuts or a hexagon head with three cuts. In indexing the work, the worm of the dividing head is thrown out of mesh and the divisions are obtained from the rapid index plate on the spindle nose.

Fig. 630a illustrates face milling, using an inserted tooth milling cutter. The cutter is mounted directly on the nose of the spindle and the longitudinal feed is used.

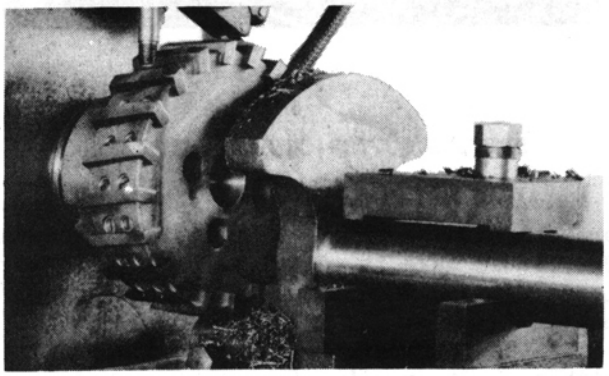


Fig. 630a

The helical milling cutter in Fig. 631 is placed as near the spindle as possible. By using plenty of cutting lubricant this heavy plain slabbing cut can be taken with a good finish.

The operation shown in Fig. 632 illustrates an excellent example of the use of the circular milling attachment in connection with the vertical spindle milling machine, for cutting circular T-slots in the saddle of a universal milling machine.

The first or plain slot is cut out on a boring mill or can be milled at the same setting shown in Fig. 632, using the two-lipped end mill, which is then replaced by a T-slot cutter.

Fig. 633 shows the method of setting a vise at 90° with an arbor. In checking, the point of the indicator is placed against the solid jaw and the longitudinal feed is used.

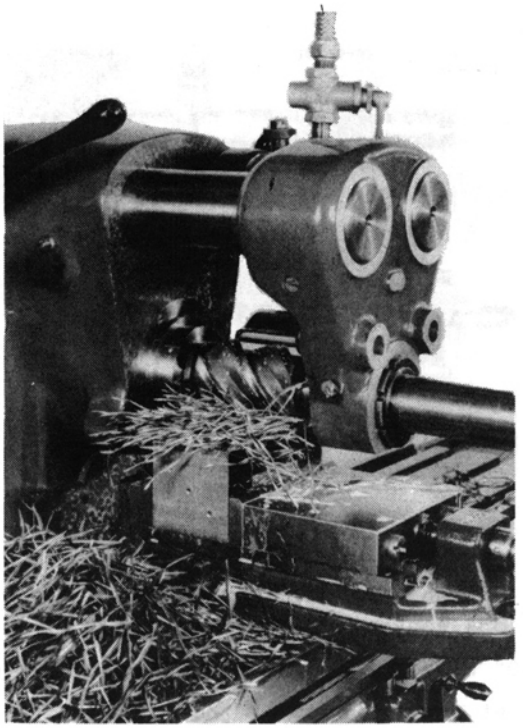


Fig. 631

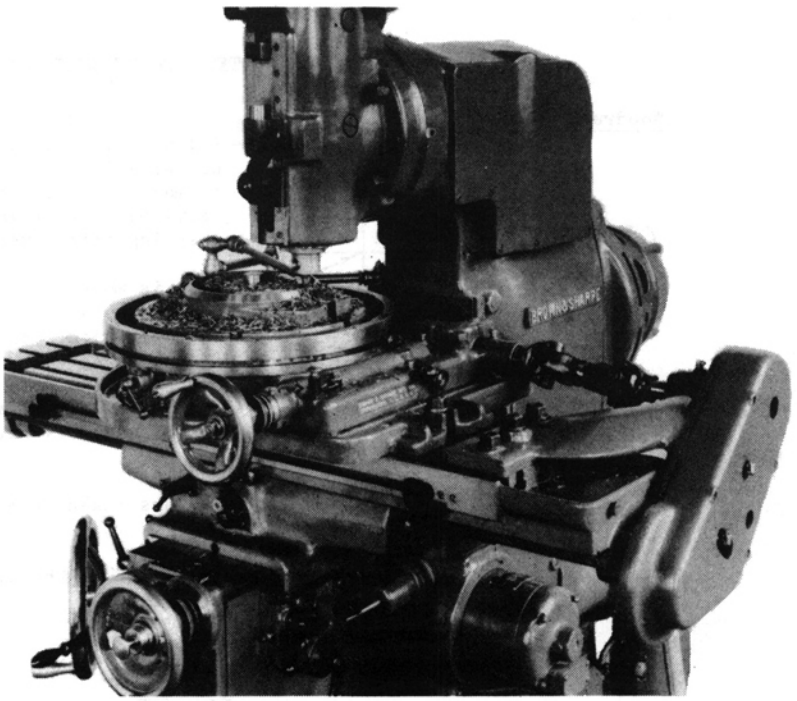


Fig. 632

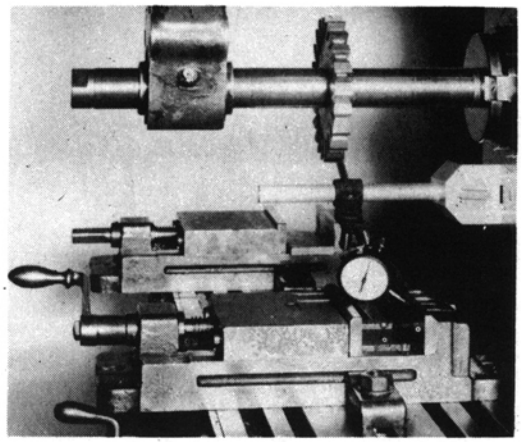


Fig. 633

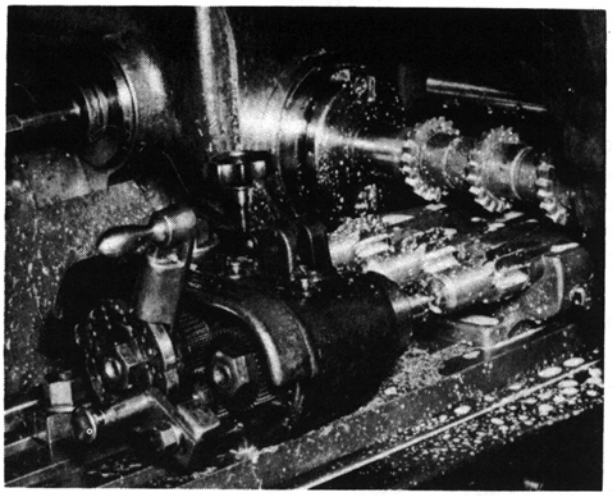


Fig. 634

The Gang Milling operation shown in Fig. 634 consists of cutting two grooves each 1.170" wide and $\frac{5}{16}$" deep in three steel forgings at one traverse of the table.

The Triple Index Centers shown in Fig. 601 are employed.

In heavy milling, the arbor must be rigidly supported by arbor yokes to prevent chattering.

A coolant should be used in milling almost all metals except cast iron.

SETTING CUTTER CENTRAL WITH WORK

In Fig. 636(a), the keyway is improperly cut. Notice that it is off-center. This would make it difficult to fit a key into the job when assembled with mating part.

Cutters are sometimes set central with a shaft by using the sight method, in which the cutter is forced into the work a few thousandths and fed across the work. A small oval, which should not exceed the width of the cutter by more than 1/16, is thus

SETTING CUTTER CENTRAL WITH WORK

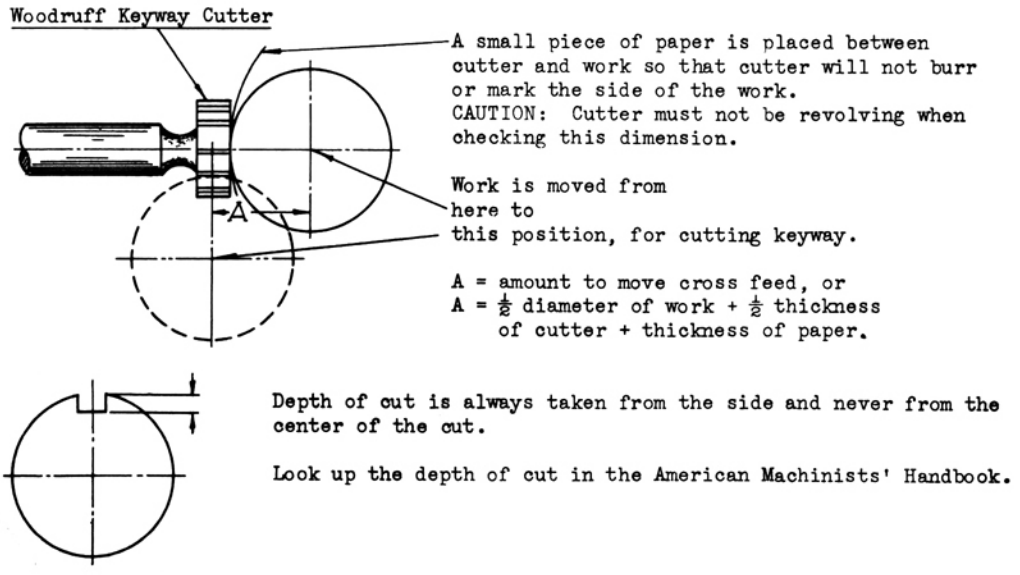
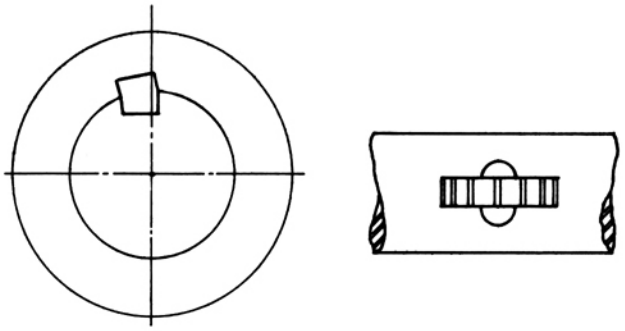
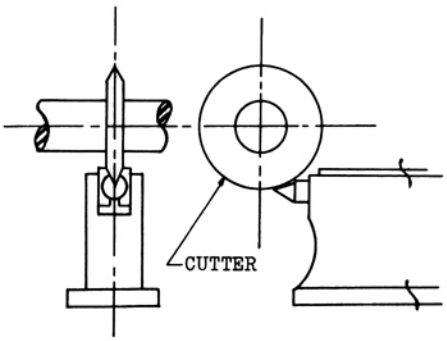

Woodruff Keyway Cutter
A small piece of paper is placed between cutter and work so that cutter will not burr or mark the side of the work.
CAUTION: Cutter must not be revolving when checking this dimension.
Work is moved from here to this position, for cutting keyway.
A = amount to move cross feed, or
A = ½ diameter of work + ½ thickness of cutter + thickness of paper.
A
Depth of cut is always taken from the side and never from the center of the cut.
Look up the depth of cut in the American Machinists' Handbook.

Fig. 635

formed. The cutter is then set so that an equal portion of the oval is visible on each side of the cutter (see Fig. 636(b)). This may be measured with a rule if desired. The cutting edge of the cutter may also be lined up with the center line of the work by sight, as shown in Fig. 636(c).



(a) (b)



(c)

Fig. 636

DIVIDING HEAD

Principle parts of the dividing head:

1. Lead screw crank
2. 2nd gear on stud
3. 1st gear on stud
4. Gear on spindle
5. Lever for locking rapid index plate
6. Angular graduations
7. Angular locating point
8. Rapid index plate
9. Pin for locking index plate
10. Index plate
11. Sector arm
12. Index crank
13. Table lock
14. Table
15. Gear on worm
16. Idler gear
17. Graduated lead screw collar
18. Position of lead screw gear

The index head (Fig. 638) is a device used to divide the periphery of a piece of work into any number of equal parts, and to hold the work in the required position while the cuts are being made. The most essential parts of the index head are the worm and worm wheel, index plates, sector arms, and change gears.

The worm wheel has 40 teeth and the worm has a single thread. The worm wheel is keyed to the spindle, and when the worm, which is turned by the index crank, is turned 40 times, the spindle is revolved once. Fractional parts of a turn are obtained by means of the index plates which are furnished with each head. The sector arms are used to mark off the number of holes on the index plate which are required to make a fractional part of a turn of the index crank, without counting them each time the index crank is moved. The

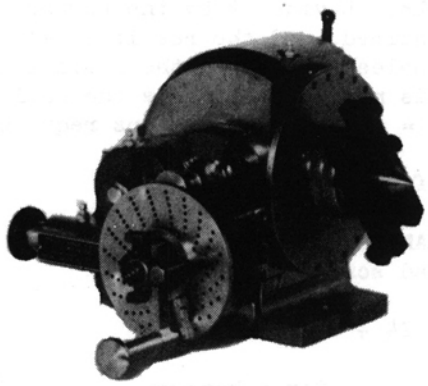


Fig. 638

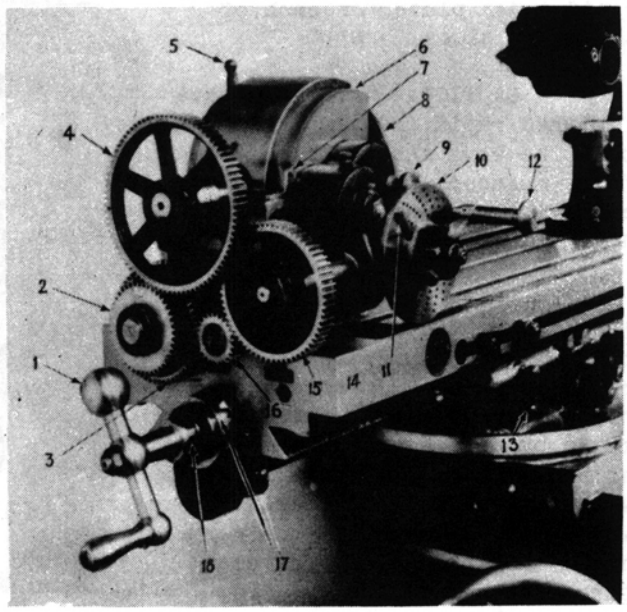


Fig. 637

change gears control the movement of the index plates and the spindle in differential indexing and helical milling. Rapid, plain, and differential indexing are the three methods most commonly used.

RAPID INDEXING. In rapid indexing, the worm and worm wheel are disengaged and the spindle is moved by hand. The required number of divisions on the work is made by means of the rapid index plate (see Fig. 638), located on the nose of the spindle. This plate usually has 24 equally spaced holes, and only the numbers which will divide evenly into 24 can be indexed (that is, 2, 3, 4, 6, 8, 12, and 24). An index pin placed in one of the holes of the rapid index plate locates the spindle in the proper position, and a clamping device locks it while the cut is being made. This method is used when a large number of duplicate parts are being milled.

RULE. Divide 24 by the number of divisions required, and the result equals the number of holes to move in the rapid index plate. This may be written as the following formula (N = number of divisions required).

$$\text{Number of Holes to Move} = \frac{24}{N}$$

EXAMPLE. Index for a hexagon head screw.

$$24 \div 6 = 4$$

To index for a hexagon head screw, machine one side and move four holes in the rapid index plate for each of the other sides.

CAUTION. After you have finished rapid indexing, see that the worm and worm wheel are fully engaged, so that the machine is again set for plain indexing.

PLAIN INDEXING. Plain indexing is a method of indexing for numbers beyond the range of rapid indexing. In this operation, the index head spindle is moved by turning an index crank attached to a worm which meshes with the worm wheel. The worm wheel has 40 teeth and the worm has a single thread. For each turn of the index crank, the worm wheel moves one tooth, or 1/40 of a revolution. To cause the spindle to make one turn, the index crank must make 40 turns, or in other words, the ratio between the revolutions of the index crank and those of the spindle is 40 to 1. A wide range of divisions may be indexed by using this method.

The number of turns or fractional parts of a turn of the index crank necessary to cut any required number of divisions may be easily determined by the following rule.

RULE. Divide 40 by the number of divisions required, and the result will equal the number of turns of the index crank. Expressed as a formula (where N = number of divisions required and T = turns of the index crank), this is:

$$\frac{40}{N} = T.$$

EXAMPLES. Index for five divisions

$$\frac{40}{N} = \frac{40}{5} = 8T.$$

Index for eight divisions

$$\frac{40}{N} = \frac{40}{8} = 5T$$

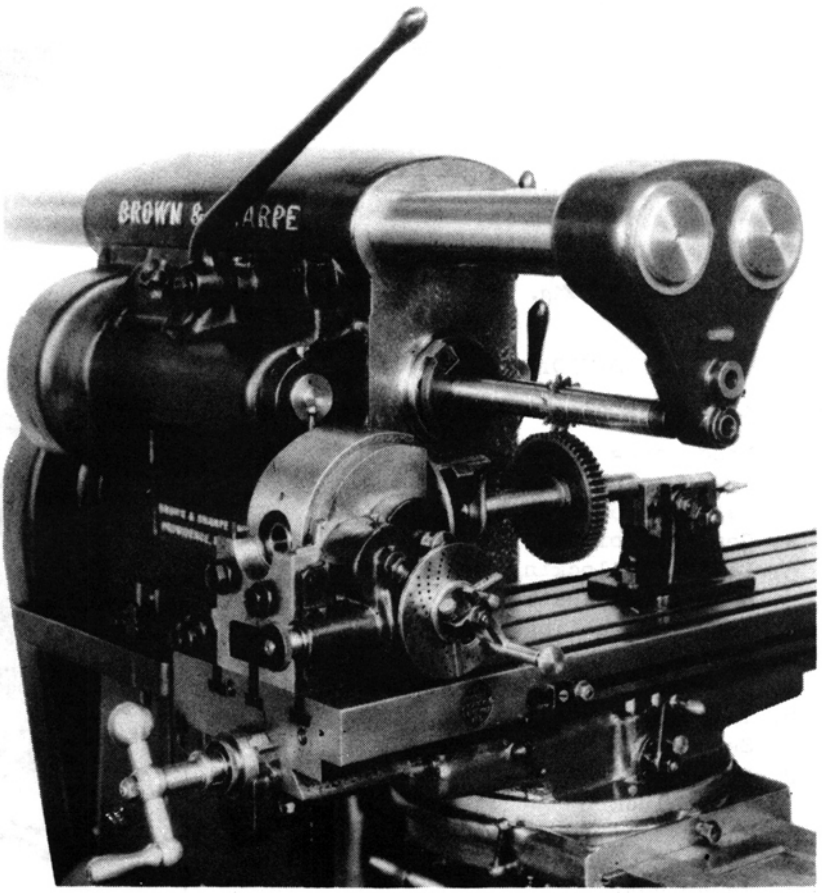


Fig. 639

Index for ten divisions

$$\frac{40}{N} = \frac{40}{10} = 4T.$$

Fig. 639 shows an index head set up for a plain indexing job.

When the number of divisions required does not divide evenly into 40, the index crank must be moved a fractional part of a turn. This is done by using index plates. The plates furnished with a Brown & Sharpe index head have circles containing the number of holes listed below. LEARN THE NUMBER OF HOLES IN EACH OF THESE CIRCLES.

Plate #1 - 15, 16, 17, 18, 19, 20
Plate #2 - 21, 23, 27, 29, 31, 33
Plate #3 - 37, 39, 41, 43, 47, 49

As another example, let us index for 18 divisions.

$$\frac{40}{N} = \frac{40}{18} = 2\frac{4}{18}T.$$

The whole number indicates the complete turns of the index crank. The denominator of the fraction represents the index circle to use and the numerator represents the number of holes to move in that circle. Since there is an index circle which contains 18 holes, we index for 18 equally spaced divisions by moving the index crank two complete turns and four holes in the 18 hole circle.

When the denominator is smaller or larger than the number of holes contained in any of the index circles, it can be increased or reduced by multiplying or dividing both terms of the fraction by a number which will give a fraction whose denominator is the same as the number of holes in one of the index plates.

For example, assume that it is necessary to index for 1-1/3 turns of the index crank. Select an index circle on one of the index plates in which the number of holes is a multiple of 3, and it will be found that either the 15, 18, 21, 27, 33, or 39 hole circle may be used. Divide the number of holes in the selected circle by 3 and obtain the common multiple for both terms of the fraction. Assuming that the 27 hole circle was selected, the common multiple would be obtained by dividing 27 by 3, giving a result of 9. Multiplying each term of the fraction by 9, we have

$$\frac{1}{3} \times \frac{9}{9} = \frac{9}{27}$$

The denominator indicates the number of holes in the circle and the numerator indicates the number of holes to move the index crank pin that circle. When the index crank pin is moved 9 holes in the 27 hole circle, 1/3 of a turn has been made.

When the number of divisions to be indexed exceeds 40, both terms of the fraction may be divided by a common divisor to obtain an index circle which is available. If 160 divisions are required, for example, the fraction to be used is

$$\frac{40}{N} \text{ or } \frac{40}{160}$$

Since there is no 160 hole circle, this fraction can be reduced to 1/4 by dividing each term of the fraction by 40. The only index circle in the standard index plates which is a multiple of 4 is the 16 hole circle, and by multiplying each term of the fraction by 4, we obtain the following result:

$$\frac{1}{4} \times \frac{4}{4} = \frac{4}{16}$$

EXAMPLES

Index for 6 divisions

$$\frac{40}{N} = \frac{40}{6} = 6\frac{2}{3}T$$

$$\frac{2}{3} \times \frac{5}{5} = \frac{10}{15} \text{ or}$$

6 turns 10 holes 15 circle

Index for 9 divisions

$$\frac{40}{N} \times \frac{40}{9} = 4\frac{4}{9}T$$

$$\frac{4}{9} \times \frac{2}{2} = \frac{8}{18} \text{ or}$$

4 turns 8 holes 18 circle

Index for 65 divisions

$$\frac{40}{N} = \frac{40}{65} = \frac{8}{13}$$

$$\frac{8}{13} \times \frac{3}{3} = \frac{24}{39} \text{ or}$$

24 holes 39 circle

Index for 136 divisions

$$\frac{40}{N} = \frac{40}{136} = \frac{5}{17}$$

$$\frac{5}{17} \times \frac{1}{1} = \frac{5}{17} \text{ or}$$

5 holes 17 circle

CAUTION. In setting the sector arms to space off the proper number of holes in the index circle, DO NOT count the hole the index crank pin is in.

DIFFERENTIAL INDEXING. The differential method of indexing is used in indexing for numbers beyond the range of plain indexing. This is accomplished by connecting the index plate to the spindle by means of a gear train, so that the index plate can rotate in relationship to the movement of the spindle. By a proper arrangement of the gearing, the index plate can be made to move fast or slow, and in the same direction (positive) or in the opposite direction (negative) to the index crank. This causes the movement of the index plate to be either greater or less than the actual movement of the index crank. Before differential indexing is attempted, gearing and the forming of gear ratios should be understood (see GEARING section).

The standard change gears (12 in all) that are furnished with each index head have the following number of teeth: 24 (two gears), 28, 32, 40, 44, 48, 56, 64, 72, 86, and 100. Special gears having 46, 47, 52, 58, 68, 70, 76, and 84 teeth may also be furnished. LEARN THE NUMBER OF TEETH IN EACH OF THESE GEARS.

When the required number of divisions cannot be indexed by plain indexing, an approximate number of divisions which can be indexed by the plain indexing method is selected. The difference between the movement of the spindle thus secured and the necessary movement is corrected by the use of change gears. The proper gearing is found in the following manner.

1st Step. Plain Indexing $-\frac{40}{n} = T$

(n = selected number)

Select some number either greater or less than the required number for which plain indexing can be used.

2nd Step. Gear Ratio - $(n - N) \times \frac{40}{n}$

The gear ratio is found by using the formula $(n - N) \times \frac{40}{n}$, in which n = approximate number and N = required number.

3rd Step. Select Gears.

The gears may be either simple or compound. State idlers to be used.

Simple Gearing

$$\frac{3}{8} \times \frac{8}{8} = \frac{24}{64}$$

Compound Gearing

$$\frac{16}{33} = \frac{\cancel{1} \times \cancel{16}}{\cancel{3} \times \cancel{11}} = \frac{\overset{24}{\cancel{24}}}{\underset{72}{\cancel{72}}} \times \frac{\overset{64}{\cancel{64}}}{\underset{44}{\cancel{44}}}$$

In differential indexing, the numerators of the fractions indicate the driving gears and the denominators indicate the driven gears. Idler gears control the direction of rotation of the index plate and are arranged as follows.

Simple Gearing - 1 idler for positive motion of index plate
2 idlers for negative motion of index plate

Compound Gearing - 1 idler for negative motion of index plate
No idlers for positive motion of index plate

In case #1 an approximate number greater than the required number was selected. Note that when the approximate number is greater than the required number, the index plate must turn in the positive direction. Using simple gearing, this requires one idler gear.

In case #2 an approximate number smaller than the required number was used. Notice that when the approximate number is less than the required number, the index plate must turn in the negative direction. Using simple gearing, two idler gears are required.

In case #3 an approximate number smaller than the required number was used. Notice that in this case compound gearing is necessary, and that since the approximate number is less than the required number, the index plate must turn in the negative direction. This requires one idler gear.

These examples show that the approximate number may be greater or less than the required number and that the speed and the direction of rotation of the index plate can be controlled by the change gears. The difference between the approximate number and the required number is limited only by the index hole circles and change gears which are available.

To index for 57 divisions:

	CASE #1	CASE #2	CASE #3
STEP #1 PLAIN INDEXING	$\frac{40}{N} = \frac{40}{57}$ $\frac{40}{60} = \frac{2}{3}$ $\frac{2}{3} \times \frac{7}{7} = \frac{14}{21}$ 14 Holes 21 Circle	$\frac{40}{N} = \frac{40}{57}$ $\frac{40}{56} = \frac{5}{7}$ $\frac{5}{7} \times \frac{3}{3} = \frac{15}{21}$ 15 Holes 21 Circle	$\frac{40}{N} = \frac{40}{57}$ $\frac{40}{54} = \frac{20}{27}$ 20 Holes 27 Circle
STEP #2 GEAR RATIO	$(n - N) \times \frac{40}{n} =$ $(60 - 57) \times \frac{40}{60} =$ $\cancel{3} \times \frac{2}{\cancel{3}} = \frac{2}{1}$	$(n - N) \times \frac{40}{n} =$ $(56 - 57) \times \frac{40}{56} =$ $-1 \times \frac{5}{7} = -\frac{5}{7}$	$(n - N) \times \frac{40}{n} =$ $(54 - 57 \times \frac{40}{54} =$ $-\cancel{3} \times \frac{20}{\cancel{27}\ 9} \quad -\frac{20}{9}$
STEP #3 SELECT GEARS	$\frac{2}{1} \times \frac{24}{24} = \frac{48}{24}$ Driver / Driven 1 Idler	$-\frac{5}{7} \times \frac{8}{8} = \frac{-40}{56}$ Driver / Driven 2 Idlers	$-\frac{20}{9} = \frac{64 \times 40}{-4 \times 5}$ Driver / $\frac{}{\cancel{3} \times \cancel{3}}$ Driven $48 \quad 24$ 1 Idler

Fig. 640 shows an index head geared for differential indexing (simple gearing) while (Fig. 641) shows an index head geared for differential indexing (compound gearing).

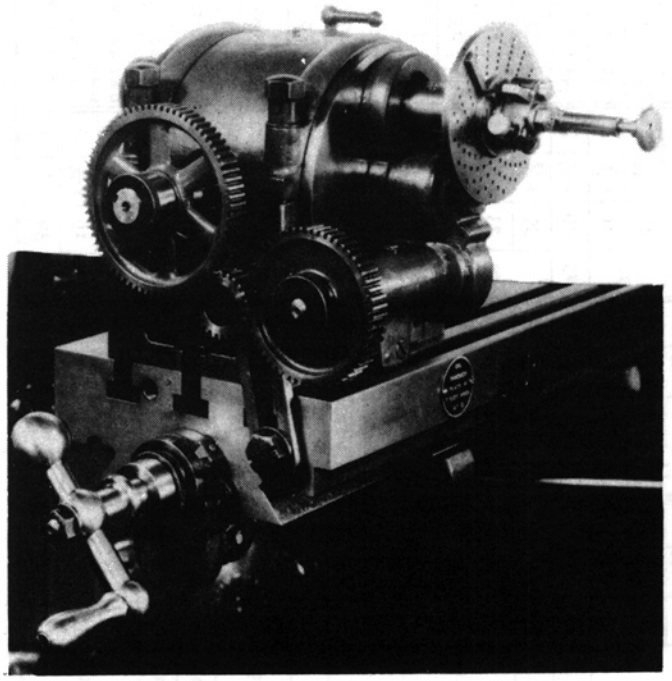


Fig. 640

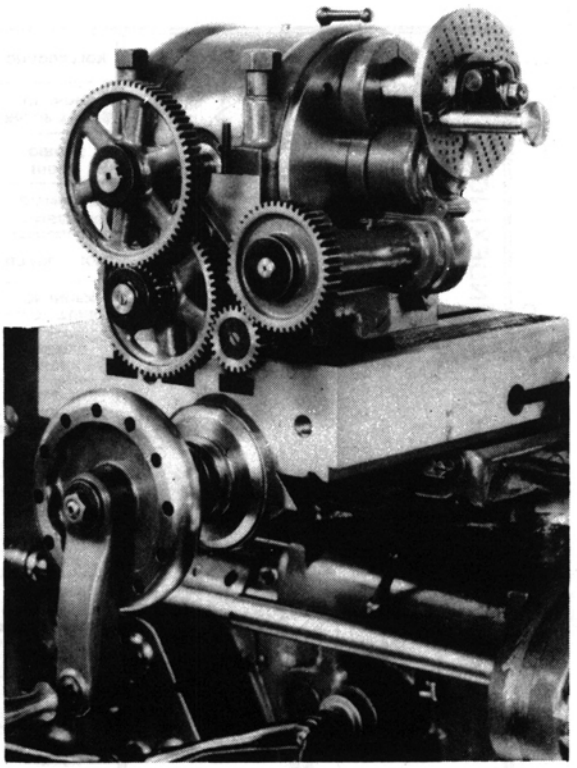
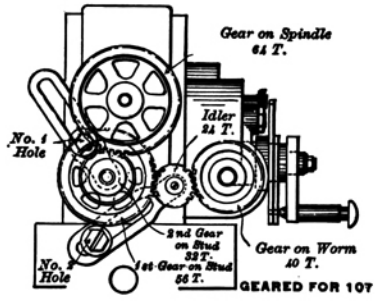


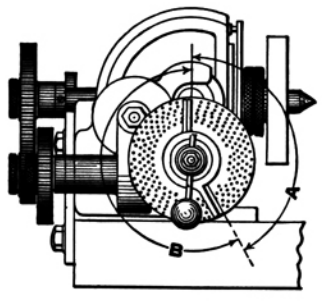
Fig. 641

A chart for plain and differential indexing, similar to the one shown below, is available in most milling machine departments.

INDEX TABLE 2 to 50

PLAIN & DIFFERENTIAL INDEXING





NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION
2	Any	20	
3	39	13 13/39	65
	33	13 11/33	65
	18	13 6/18	65
4	Any	10	
5	Any	8	
6	39	6 26/39	132
	33	6 22/33	132
	18	6 12/18	132
7	49	5 35/49	140
	21	5 15/21	142
8	Any	5	
9	27	4 12/27	88
	18	4 8/18	87
10	Any	4	
11	33	3 22/33	126
12	39	3 13/39	65
	33	3 11/33	65
	18	3 6/18	65

NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION
13	39	3 3/39	14
14	49	2 42/49	169
	21	2 18/21	170
15	39	2 26/39	132
	33	2 22/33	132
	18	2 12/18	132
16	20	2 10/20	98
17	17	2 6/17	69
18	27	2 6/27	43
	18	2 4/18	43
19	19	2 2/19	19
20	Any	2	
21	21	1 19/21	18*
22	33	1 27/33	161
23	23	1 17/23	147
24	39	1 26/39	132
	33	1 22/33	132
	18	1 12/18	132
25	20	1 12/20	118

NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION
26	39	1 21/39	106
27	27	1 13/27	95
28	49	1 21/49	83
	21	1 9/21	85
29	29	1 11/29	75
30	39	1 13/39	65
	33	1 11/33	65
	18	1 6/18	65
31	31	1 9/31	56
32	20	1 5/20	48
33	33	1 7/33	41
34	17	1 3/17	33
35	49	1 7/49	26
	21	1 3/21	28
36	27	1 3/27	21
	18	1 2/18	21
37	37	1 3/37	15
38	19	1 1/19	9
39	39	1 1/39	3

NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION
40	Any	1	
41	41	40/41	3*
42	21	20/21	9*
43	43	40/43	12*
44	33	30/33	17*
45	27	24/27	21*
	18	16/18	21*
46	23	20/23	172
47	47	40/47	168
48	18	15/18	165
49	49	40/49	161
50	20	16/20	158

GRADUATIONS IN TABLE INDICATE SETTING FOR ARMS OF SECTOR WHEN INDEX CRANK MOVES THROUGH ARC "A," EXCEPT CASES MARKED * WHEN THE INDEX CRANK MOVES THROUGH ARC "B."

INDEX TABLE 51 to 92.

NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION	GEAR ON WORM	No.1 HOLE		GEAR ON SPINDLE	IDLERS	
					1ST GEAR ON STUD	2ND GEAR ON STUD		No. 1 HOLE	No. 2 HOLE
51	17	14/17	33*	24			48	24	44
52	39	30/39	152						
53	49	35/49	140	56	40	24	72		
	21	15/21	142	56	40	24	72		
54	27	20/27	147						
55	33	24/33	144						
56	49	35/49	140						
	21	15/21	142						
57	49	35/49	140	56			40	24	44
	21	15/21	142	56			40	24	44
58	29	20/29	136						
59	39	26/39	132	48			32	44	
	33	22/33	132	48			32	44	
	18	12/18	132	48			32	44	
60	39	26/39	132						
	33	22/33	132						
	18	12/18	132						
61	39	26/39	132	48			32	24	44
	33	22/33	132	48			32	24	44
	18	12/18	132	48			32	24	44
62	31	20/31	127						
63	39	26/39	132	24			48	24	44
	33	22/33	132	24			48	24	44
	18	12/18	132	24			48	24	44
64	16	10/16	123						
65	39	24/39	121						
66	33	20/33	120						
67	49	28/49	112	28			48	44	
	21	12/21	113	28			48	44	
68	17	10/17	116						

NUMBER OF DIVISIONS	INDEX CIRCLE	No. OF TURNS OF INDEX	GRADUATION	GEAR ON WORM	No.1 HOLE		GEAR ON SPINDLE	IDLERS	
					1ST GEAR ON STUD	2ND GEAR ON STUD		No. 1 HOLE	No. 2 HOLE
69	20	12/20	118	40			56	24	44
70	49	28/49	112						
	21	12/21	113						
71	27	15/27	110	72			40	24	
	18	10/18	109	72			40	24	
72	27	15/27	110						
	18	10/18	109						
73	49	28/49	112	28			48	24	44
	21	12/21	113	28			48	24	44
74	37	20/37	107						
75	15	8/15	105						
76	19	10/19	103						
77	20	10/20	98	32			48	44	
78	39	20/39	101						
79	20	10/20	98	48			24	44	
80	20	10/20	98						
81	20	10/20	98	48			24	24	44
82	41	20/41	96						
83	26	10/20	98	32			48	24	44
84	21	10/21	94						
85	17	8/17	92						
86	43	20/43	91						
87	15	7/15	92	40			24	24	44
88	33	15/33	89						
89	27	12/27	88	72			32	44	
	18	8/18	87	72			32	44	
90	27	12/27	88						
	18	8/18	87						
91	39	18/39	91	24			48	24	44
92	23	10/23	86						

EXERCISES

1. Briefly describe the index head and explain its purpose.
2. Describe the following parts of an index head and explain the purpose of each: (a) Worm and worm wheel, (b) Index plates, (c) Sector arms, and (d) change gears.
3. Tell what the usual index head ratio is, and explain what it means.
4. Name and explain briefly three kinds of indexing in common use.
5. Explain how the sector arms should be set for indexing twelve divisions.
6. Explain what must be done in changing from rapid indexing to plain indexing.
7. Explain what must be done to change the index head from plain indexing to differential indexing.
8. List the number of holes in each of the standard index plates, and the number of teeth contained in the standard change gears furnished with a Brown & Sharpe index head.
9. Explain the meaning of the following terms: (a) Gears, (b) Gear ratio, (c) Gear train, (d) Idler gear, (e) Simple gear ratio, and (f) Compound gear ratio.
10. Index for the following equally spaced divisions: 8, 12, 24, 37, 43, 56, 61, 96, 129, 173.

GRADUATING

Flat rules and verniers may be graduated (divided into regular intervals) on a milling machine, by using a pointed tool and an index head. The tool is held stationary in a fly cutter holder (Fig. 642). This is mounted in the spindle of the machine, or it may be fastened to the spindle of a vertical milling machine or rack cutting attachment. The work is clamped to the table parallel to the "T" slots. The index head spindle is geared to the table feed screw with gears having a 1-1 ratio. The table is moved longitudinally by turning the index crank. Fractional parts of a turn are obtained by means of the index plates, the same as in plain indexing. The lines are cut by moving the table transversely under the point of the tool. The movement of the table is controlled with the hand feeds.

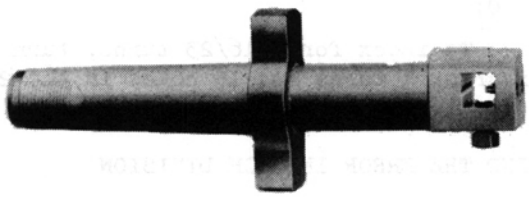


Fig. 642

Fig. 643 shows the milling machine set-up for graduating. Notice how the gears are arranged. When the index crank is turned one revolution, the spindle turns 1/40 of a revolution, and through the 1-1 ratio causes the lead screw to move 1/40 of a revolution. Since the usual lead of the lead screw is .250, one turn of the index crank will move the table.

$$\frac{1}{40} \text{ of } \frac{.250}{1} = .00625.$$

When the table is to be moved any required distance, divide the required distance by the distance advanced in one turn of the index crank, and the result will be the number of turns of the index crank that are necessary. This may be written as a formula, as follows:

$$T = \frac{W}{.00625}$$

where T = number of turns of index crank, W = width of divisions, and .00625 = the distance that the table moves in one turn of the index crank.

SHOW HOW TO INDEX FOR LINES 1/32 (.03125) APART

$$\frac{W}{.00625} = \frac{.03125}{.00625} = 5$$

Turn the index crank five turns in any circle.

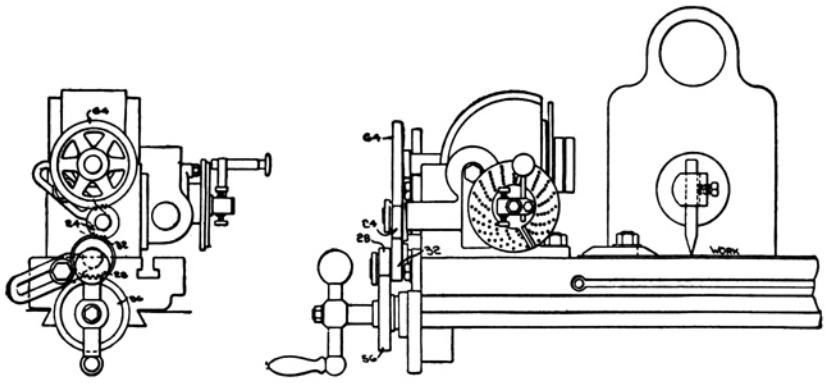


Fig. 643

For instructions in combining fractions and continued fractions see Practical Shop Mathematics, Volume II--Advanced, by Wolfe and Phelps, McGraw-Hill Book Co.

SHOW HOW TO INDEX FOR LINES .0481 APART

$\frac{W}{.00625} = \frac{.04810}{.00625} = 7\frac{435}{625}$ turns of the index crank

$7\frac{435}{625}$ may be changed to $7\frac{16}{23}$ by continued fractions, as follows:

435	625	1
380	435	2
55	190	3
50	165	2
5	25	5
	25	
	0	

		1	2	3	2	5		
1	0	1	2	7	16	87	x 5 =	435
0	1	1	3	10	23	125	x 5 =	625
		1	2	3	2	5		

To index for 7-16/23 turns, turn the index crank 7 turns and 16 holes in the 23 hole circle.

TO FIND THE ERROR IN EACH DIVISION

The error in each division is equal to the difference between the distance the table should move and the distance the selected indexing actually moves it.

$\frac{87}{125} \times \frac{.00625}{1} = .0043500$ distance the table should move

$\frac{16}{23} \times \frac{.00625}{1} = .0043478$ distance the table actually moves

$.0000022$ error in each division

TO FIND THE ERROR IN EACH INCH

The error in each inch is equal to the number of divisions in one inch multiplied by the error in each division.

1.000 ÷ W = number of divisions in 1"

1.000 ÷ .0481 = 20.79 divisions in 1"

Error in each division X divisions in 1" = error in 1"

.0000022 X 20.79 = 0.000045 error in 1"

USE OF ANGULAR INDEXING TABLE

60 Seconds (") = 1'
60 Minutes (') = 1°

90° (Degrees) = 1 Right Angle
360° (Degrees) = 1 Circle

The index head requires 40 turns of the index crank to move the spindle through one revolution, or 360°. Therefore one turn of the index crank will move the spindle 1/40 x 360° = 9°.

9° = 9 x 60' or 540'
9° = 9 x 60' x 60" or 32,400"

In angles given only in degrees, divide the required number of degrees by 9 to obtain the required turns of the index crank. In angles involving degrees and minutes, reduce the angles to minutes and divide by 540. In angles involving degrees, minutes, and seconds, reduce the angles to seconds and divide by 32,400. Carry the division to the fourth decimal place, and the result will be the number of turns of the index crank necessary to index the angle.

To find the index circle which must be used for the decimal part of a turn, find the required decimal or the nearest decimal to it in the table on page 175. The index circle is listed under "C" and the number of holes to move in that circle is listed under "H". The following examples show application of the table.

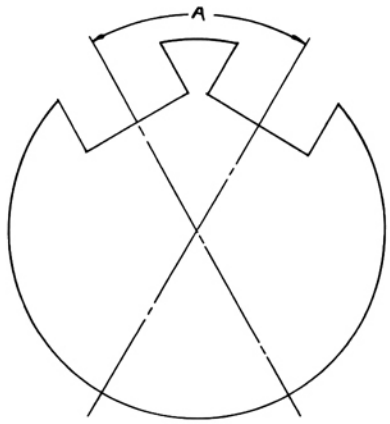


Fig. 644

1. Angle A = 24°45'

24° = 24 X 60' = 1440'
45' = 45'
1485'

1485 ÷ 540 = 2.7500

In the table opposite .7500, 12 is under "H" and 16 is under "C". TOTAL INDEXING = 2 Turns, 12 Holes, 16 Circle.

ANGULAR INDEXING TABLE

Value	H	C	Value	H	C	Value	H	C	Value	H	C	Value	H	C	Value	H	C	Value	H	C	Value	H	C
.0204	1	49	.1395	6	43	.2609	6	23	.3846	15	39	.5106	24	47	.6279	27	43	.7500	12	16	.8718	34	39
.0213	1	47	.1429	3	21	.2632	5	19	.3871	12	31	.5116	22	43	.6296	17	27	.7500	15	20	.8723	41	47
.0233	1	43	.1429	7	49	.2653	13	49	.3878	19	49	.5122	21	41	.6316	12	19	.7551	37	49	.8750	14	16
.0244	1	41	.1463	6	41	.2667	4	15	.3888	7	18	.5128	20	39	.6326	31	49	.7561	31	41	.8776	43	49
.0256	1	39	.1481	4	27	.2683	11	41	.3902	16	41	.5135	19	37	.6341	26	41	.7568	28	37	.8780	36	41
.0270	1	37	.1489	7	47	.2703	10	37	.3913	9	23	.5151	17	33	.6364	21	33	.7576	25	33	.8788	29	33
.0303	1	33	.1500	3	20	.2727	9	33	.3939	13	33	.5161	16	31	.6383	30	47	.7586	22	29	.8824	15	17
.0323	1	31	.1515	5	33	.2759	8	29	.3953	17	43	.5172	15	29	.6410	25	39	.7619	16	21	.8837	38	43
.0345	1	29	.1538	6	39	.2766	13	47	.4000	6	15	.5185	14	27	.6452	20	31	.7647	13	17	.8888	16	18
.0370	1	27	.1579	3	19	.2777	5	18	.4000	8	20	.5217	12	23	.6471	11	17	.7674	33	43	.8888	24	27
.0408	2	49	.1613	5	31	.2791	12	43	.4043	19	47	.5238	11	21	.6486	24	37	.7692	30	39	.8919	33	37
.0426	2	47	.1622	6	37	.2821	11	39	.4054	15	37	.5263	10	19	.6500	13	20	.7742	24	31	.8936	42	47
.0435	1	23	.1628	7	43	.2857	14	49	.4074	11	27	.5294	9	17	.6512	28	43	.7755	38	49	.8947	17	19
.0465	2	43	.1633	8	49	.2857	6	21	.4082	20	49	.5306	26	49	.6522	15	23	.7760	36	47	.8966	26	29
.0476	1	21	.1666	3	18	.2903	9	31	.4103	16	39	.5319	25	47	.6531	32	49	.7777	21	27	.8974	35	39
.0488	2	41	.1702	8	47	.2927	12	41	.4118	7	17	.5333	8	15	.6552	19	29	.7777	14	18	.8980	44	49
.0500	1	20	.1707	7	41	.2941	5	17	.4138	12	29	.5349	23	43	.6585	27	41	.7805	32	41	.9000	18	20
.0513	2	39	.1724	5	29	.2963	8	27	.4146	17	41	.5366	22	41	.6596	31	47	.7826	18	23	.9024	37	41
.0526	1	19	.1739	4	23	.2973	11	37	.4186	18	43	.5385	21	39	.6666	10	15	.7838	29	37	.9032	28	31
.0541	2	37	.1765	3	17	.2979	14	47	.4194	13	31	.5405	20	37	.6666	12	18	.7872	37	47	.9048	19	21
.0555	1	18	.1795	7	39	.3000	6	20	.4211	8	19	.5454	18	33	.6666	14	21	.7879	26	33	.9070	39	43
.0588	1	17	.1818	6	33	.3023	13	43	.4242	14	33	.5484	17	31	.6666	18	27	.7895	15	19	.9090	30	33
.0606	2	33	.1837	9	49	.3030	10	33	.4255	20	47	.5500	11	20	.6666	22	33	.7907	34	43	.9130	21	23
.0612	3	49	.1852	5	27	.3043	7	23	.4286	9	21	.5510	27	49	.6666	26	39	.7931	23	29	.9149	43	47
.0625	1	16	.1860	8	43	.3061	15	49	.4286	21	49	.5517	16	29	.6735	33	49	.7949	31	39	.9184	45	49
.0638	3	47	.1875	3	16	.3077	12	39	.4324	16	37	.5532	26	47	.6744	29	43	.7959	39	49	.9189	34	37
.0645	2	31	.1892	7	37	.3103	9	29	.4348	10	23	.5555	10	18	.6757	25	37	.8000	12	15	.9231	36	39
.0666	1	15	.1905	4	21	.3125	5	16	.4359	17	39	.5555	15	27	.6774	21	31	.8000	16	20	.9259	25	27
.0690	2	29	.1915	9	47	.3158	6	19	.4375	7	16	.5581	24	43	.6809	32	47	.8049	33	41	.9268	38	41
.0698	3	43	.1935	6	31	.3171	13	41	.4390	18	41	.5610	23	41	.6829	28	41	.8065	25	31	.9302	40	43
.0732	3	41	.1951	8	41	.3191	15	47	.4419	19	43	.5625	9	16	.6842	13	19	.8085	38	47	.9310	27	29
.0741	2	27	.2000	3	15	.3226	10	31	.4444	8	18	.5641	22	39	.6875	11	16	.8095	17	21	.9333	14	15
.0769	3	39	.2000	4	20	.3243	12	37	.4444	12	27	.5652	13	23	.6897	20	29	.8108	30	37	.9355	29	31
.0811	3	37	.2041	10	49	.3256	14	43	.4468	21	47	.5676	21	37	.6923	27	39	.8125	13	16	.9362	44	47
.0816	4	49	.2051	8	39	.3265	16	49	.4483	13	29	.5714	12	21	.6939	34	49	.8140	35	43	.9375	15	16
.0851	4	47	.2069	6	29	.3333	5	15	.4490	22	49	.5714	28	49	.6957	16	23	.8148	22	27	.9388	46	49
.0870	2	23	.2093	9	43	.3333	6	18	.4500	9	20	.5745	27	47	.6969	23	33	.8163	40	49	.9394	31	33
.0909	3	33	.2105	4	19	.3333	7	21	.4516	14	31	.5757	19	33	.6977	30	43	.8181	27	33	.9412	16	17
.0930	4	43	.2121	7	33	.3333	9	27	.4545	15	33	.5789	11	19	.7000	14	20	.8205	32	39	.9444	17	18
.0952	2	21	.2128	10	47	.3333	11	33	.4595	17	37	.5806	18	31	.7021	33	47	.8235	14	17	.9459	35	37
.0968	3	31	.2162	8	37	.3333	13	39	.4615	18	39	.5814	25	43	.7027	26	37	.8261	19	23	.9474	18	19
.0976	4	41	.2174	5	23	.3404	16	47	.4634	19	41	.5854	24	41	.7037	19	27	.8276	24	29	.9487	37	39
.1000	2	20	.2195	9	41	.3415	14	41	.4651	20	43	.5862	17	29	.7059	12	17	.8293	34	41	.9500	19	20
.1020	5	49	.2222	6	27	.3448	10	29	.4667	7	15	.5882	10	17	.7073	29	41	.8298	39	47	.9512	39	41
.1026	4	39	.2222	4	18	.3469	17	49	.4681	22	47	.5897	23	39	.7097	22	31	.8333	15	18	.9524	20	21
.1034	3	29	.2245	11	49	.3478	8	23	.4694	23	49	.5918	29	49	.7143	15	21	.8367	41	49	.9535	41	43
.1053	2	19	.2258	7	31	.3488	15	43	.4706	8	17	.5926	16	27	.7143	35	49	.8372	36	43	.9565	22	23
.1064	5	47	.2308	9	39	.3500	7	20	.4737	9	19	.5946	22	37	.7179	28	39	.8378	31	37	.9574	45	47
.1081	4	37	.2326	10	43	.3514	13	37	.4762	10	21	.5957	28	47	.7209	31	43	.8387	26	31	.9592	47	49
.1111	2	18	.2340	11	47	.3529	6	17	.4783	11	23	.6000	9	15	.7222	13	18	.8421	16	19	.9630	26	27
.1111	3	27	.2353	4	17	.3548	11	31	.4815	13	27	.6000	12	20	.7234	34	47	.8462	33	39	.9655	28	29
.1163	5	43	.2381	5	21	.3590	14	39	.4828	14	29	.6047	26	43	.7241	21	29	.8485	28	33	.9677	30	31
.1176	2	17	.2414	7	29	.3617	17	47	.4839	15	31	.6060	20	33	.7273	24	33	.8500	17	20	.9697	32	33
.1212	4	33	.2424	8	33	.3636	12	33	.4848	16	33	.6087	14	23	.7297	27	37	.8511	40	47	.9730	36	37
.1220	5	41	.2432	9	37	.3659	15	41	.4865	18	37	.6098	25	41	.7317	30	41	.8519	23	27	.9744	38	39
.1224	6	49	.2439	10	41	.3673	18	49	.4872	19	39	.6111	11	18	.7333	11	15	.8537	35	41	.9756	40	41
.1250	2	16	.2449	12	49	.3684	7	19	.4878	20	41	.6122	30	49	.7347	36	49	.8571	18	21	.9767	42	43
.1277	6	47	.2500	4	16	.3704	10	27	.4884	21	43	.6129	19	31	.7368	14	19	.8571	42	49	.9787	46	47
.1282	5	39	.2500	5	20	.3721	16	43	.4894	23	47	.6154	24	39	.7391	17	23	.8605	37	43	.9796	48	49
.1290	4	31	.2553	12	47	.3750	6	16	.4898	24	49	.6170	29	47	.7407	20	27	.8621	25	29			
.1304	3	23	.2558	11	43	.3784	14	37	.5000	8	16	.6190	13	21	.7419	23	31	.8649	32	37			
.1333	2	15	.2564	10	39	.3793	11	29	.5000	9	18	.6207	18	29	.7436	29	39	.8666	13	15			
.1351	5	37	.2581	8	31	.3810	8	21	.5000	10	20	.6216	23	37	.7442	32	43	.8696	20	23			
.1379	4	29	.2593	7	27	.3830	18	47	.5102	25	49	.6250	10	16	.7447	35	47	.8710	27	31			

Examples showing the use of this table are given on pages 174 and 176.

2. Angle A = 16°26'

$18^\circ = 18 \times 60' = 1080'$
$26' = \underline{\quad 26'}$
$1106'$

$1106 \div 540 = 2.0481$

Required decimal = .0481
Nearest decimal = .0476
.0005

$540 \times .0005 = .27'$
$.27 \times 60" = 16.2"$ ERROR

In the table opposite .0476, 1 is under "H" and 21 is under "C". TOTAL INDEXING = 2 Turns, 1 Hole, 21 Circle.

3. Angle A = 24°54'23"

$24^\circ = 24 \times 60' \times 60" = 86{,}400"$
$54' = 54 \times 60" = 3{,}240"$
$23" = \underline{\quad 23"}$
$89{,}663"$

$89{,}663 \div 32{,}400 = 2.7674$

In the table opposite .7674, 33 is under "H" and 43 is under "C". TOTAL INDEXING = 2 Turns, 33 Holes, 43 Circle.

4. Angle A = 39°51'21"

$39^\circ = 39 \times 60' \times 60" = 140{,}400"$
$51^\circ = 51 \times 60" = 3{,}060"$
$21" = \underline{\quad 21"}$
$143{,}481"$

$143{,}481 \div 32{,}400 = 4.4284$

Nearest decimal = .4286
Required decimal = .4284
.0002

$32{,}400" \times .0002 = 6.48"$ ERROR

In the table opposite .4286, 9 is under "H" and 21 is under "C". TOTAL INDEXING = 4 Turns, 9 Holes, 21 Circle.

ANGULAR INDEXING. Angular indexing is the type of indexing used when the measurement on the job to be indexed is given as an angle. If the work requires a greater degree of accuracy than can be obtained by use of the Angular Indexing Table, differential indexing may be used.

EXAMPLES

Index for 25° (N = degrees in given angle)

$\frac{N^\circ}{9^\circ} = T \qquad \frac{N^\circ}{9^\circ} = \frac{25^\circ}{9^\circ} = 2\frac{7}{9}T \qquad 2\frac{7}{9} \times \frac{2}{2} = 2\frac{14}{18}T$

Index 2 turns and 14 holes in 18 hole circle.

Index for 12°12'. (N' = minutes in given angles)

$\frac{N'}{540'} = T$

$12^\circ = 12 \times 60' = 720'$
$12' = \underline{\quad 12'}$
$732'$

$\frac{732}{540} = 1\frac{192}{540} = 1\frac{1}{3}$ approx. $\qquad 1\frac{1}{8} \times \frac{5}{5} = 1\frac{5}{15}T$

Index 1 turn and 5 holes in 15 hole circle.

Gearing.

$1\frac{1}{3} = \frac{4}{3} \qquad \frac{122}{3} \times \frac{40}{1} = \frac{122}{3} = 40\frac{2}{3}$ turns

$40\frac{2}{3} - 40 = \frac{2}{3}$ Gear Ratio

$\frac{2}{3} \times \frac{24}{24} = \frac{48}{72}$ Driving Gear / Driven Gear $-$ 1 Idler

Index for 29°25'16". Check error in arc. (N" = seconds in given angle)

$\frac{N"}{32400"} = T$

$29^\circ = 29 \times 60' \times 60" = 104400$
$25' = 25' \times 60" = 1500$
$16" = \underline{\quad 16}$
105916

$\frac{105916}{32400} = 3\frac{3}{16}$ turns approx., or 3 turns and 3 holes in 16 hole circle.

Gearing.

$3\frac{3}{16} = \frac{51}{16} \qquad \frac{105916}{32400} \times \frac{16}{51} \times \frac{40}{1} = 41\frac{473}{20655}$

$41\frac{473}{20655} - 40 = 1\frac{473}{20655} \qquad 1\frac{473}{20655} = \frac{21128}{20655}$

20655	21128 | 1
20339	20655 | 43
316	473 | 1
314	316 | 2
2	157 | 78
2	156 | 2
	1 |

		1	43	1	2	78	2
0	1	1	44	45	134	10497	21128
1	0	1	43	44	131	10262	20655
		1	43	1	2	78	2

$$\frac{45}{44} = \frac{\cancel{9}^{\,8} \times \cancel{5}^{\,11}}{\cancel{11}^{\,1} \times \cancel{11}^{\,11}} = \frac{72}{64} \times \frac{40}{44}$$ Driving Gear / Driven Gear

Check for Error.

$$41\frac{473}{20655} = 41\frac{1}{44} \text{ approx.}$$

$$\frac{1805}{44} \times \frac{1}{40} \times \frac{51}{16} \times \frac{32400"}{1} = 105915.55"$$

105916" - 105915.55" = .45" error.

HELICAL AND SPIRAL MILLING. When the spindle of an index head is geared to the lead screw of a milling machine so that the work revolves on its axis as the table moves along the ways, a helical or spiral cut is produced. When the cut is made on cylindrical work it is called a helical cut and when made on a tapered piece it is called a spiral cut. Helical milling cutters, helical gears, twist drills, counterbores, and similar work are produced in this way.

Before a helical cut can be made, the lead of the helix, the angle of the helix, and the diameter of the work must be known. The lead is equal to the distance the table advances when the work makes one revolution. Any change in the gearing connecting the index head spindle and the lead screw will change the lead of the helix. The helix angle is the angle the cut makes with the axis of the work, and changes with any change in the lead or in the diameter of the work. THE TABLE MUST BE SET AT THE HELIX ANGLE.

The index head spindle is geared to the lead screw of the table by means of a train of change gears, as shown in Fig. 645. These gears are called the gear on the screw (D), the first gear on the stud (C), the second gear on the stud (B), and the gear on the worm (A). The gear on the screw and the first gear on the stud are the driving gears and the second gear on the stud and the gear on the worm are the driven gears. This may be expressed as a ratio:

$$\frac{\text{Driven Gears}}{\text{Driving Gears}} = \frac{\text{2nd} \times \text{Worm}}{\text{1st} \times \text{Screw}} = \frac{A \times B}{C \times D}$$

By using different combinations of change gears, the distance that the table moves while the spindle revolves once may be changed. In other words, the lead that is cut depends directly on the gears that are used. Usually (though not always) the gear ratio is such that the work is advanced more than one inch while it makes one revolution. Therefore, the lead is expressed in inches per revolution rather than in revolutions per inch, as in threads. For example, a helix is said to have an eight inch lead rather than that its pitch is one-eighth turn per inch.

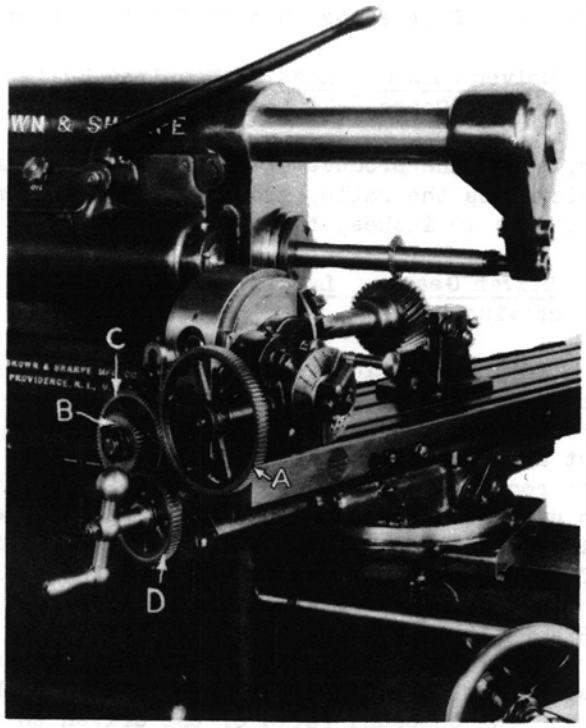


Fig. 645

The table feed screw usually has four threads per inch and a lead of one-fourth of an inch. Motion is transferred from the lead screw to the spindle through the worm and worm wheel, which have a 40 to 1 ratio. When the spindle makes one revolution, the table moves ten inches along the ways if even gearing (1 to 1 ratio) is used. One revolution of spindle x Index head ratio x Gear ratio x Lead of lead screw = Lead of machine, or

$$1 \times \frac{40}{1} \times \frac{1}{1} \times \frac{1"}{4} = \text{Lead of machine.}$$

The standard lead of a milling machine is 10" and all change gears are figured on this basis. Any change in gear ratio makes a corresponding change in the lead.

EXAMPLES

(1) Using 1 to 4 ratio

$$1 \times \frac{\cancel{40}^{\,5}}{1} \times \frac{1}{\cancel{4}^{\,2}} \quad \frac{1"}{\cancel{4}^{\,2}} = \frac{5}{2} = 2.500" \text{ Lead}$$

(2) Using 2 to 1 ratio

$$1 \times \frac{\overset{10}{\cancel{40}}}{1} \quad \frac{2}{1} \quad \frac{1"}{\cancel{4}} = 20" \text{ Lead.}$$

The compound ratio of the driven to the driving gears equals the ratio of the lead of the required helix to the lead of the machine. Expressing this in fraction form:

$$\frac{\text{Driven Gears}}{\text{Driving Gears}} = \frac{\text{Lead of Required Helix}}{\text{Lead of Machine}}$$

Or, since the product of each class of gears determines the ratio, and the lead of the machine is 10 inches, the

$$\frac{\text{Driven Gears}}{\text{Driving Gears}} = \frac{\text{Lead of Required Helix}}{10}$$

The compound ratio of the driven to the driving gears may always be represented by a fraction whose numerator is the lead to be cut and whose denominator is 10. That is, if the required lead is 20, the ratio is 20:10. To express this in units instead of tens, divide both terms of the ratio by 10. This is often a convenient way to think of the ratio, a lead of 40 giving a ratio of 4:1, a lead of 25 a ratio of 2.5:1, etc.

To illustrate the usual calculations, assume that a helix of 12 inch lead is to be cut. The compound ratio of the driven to the driving gears equals the desired lead divided by 10, or it may be represented by the fraction 12/10. Resolving this into two factors to represent the two pairs of change gears,

$$\frac{12}{10} = \frac{3}{2} \times \frac{4}{5}$$

Both terms of the first factor are multiplied by a number (24 in this case) that will make the resulting numerator and denominator correspond with the number of teeth of two of the change gears furnished with the machine (such multiplications not affecting the value of a fraction).

$$\frac{3}{2} \times \frac{24}{24} = \frac{72}{48}$$

Treating the second factor similarly,

$$\frac{4}{5} \times \frac{8}{8} = \frac{32}{40}$$

Selecting 72, 32, 48, and 40 teeth gears,

$$\frac{12}{10} = \left(\frac{72 \times 32}{48 \times 40}\right)$$

The numerators of the fractions represent the driven gears, and the denominators the driving gears. The 72 teeth gear is the worm gear, the 40 is 1st on the stud, the 32 is 2nd on the stud, and the 48 is the screw gear. The two driven gears or the two driving gears may be transposed without changing the helix. That is, the 72 teeth gear could be used as the 2nd on the stud and the 32 teeth gear could be used as the worm gear, if desired. A third combination could also be made.

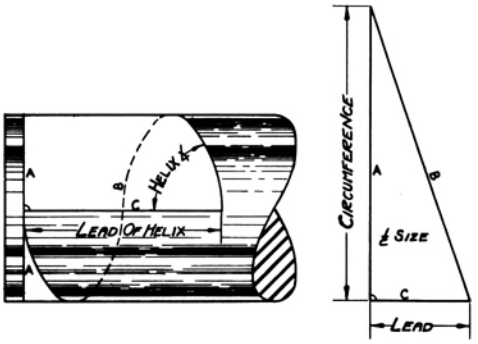


Fig. 646

Determine the gears to be used in cutting a lead of 27 inches.

$$\frac{27}{10} = \frac{3}{2} \times \frac{9}{5} = \left(\frac{3}{2} \times \frac{16}{16}\right) \times \left(\frac{9}{5} \times \frac{8}{8}\right) = \frac{48}{32} \times \frac{72}{40}$$

Determine the lead that would be cut by the gears, with 48, 72, 32, and 40 teeth, the first two being used as the driven gears.

$$\text{Helix to be cut} = \frac{10 \times 48 \times 72}{32 \times 40} = 27 \text{ inches to one revolution}$$

The milling machine table must always be set to the angle of the job.

The angle of the helix depends upon the lead of the helix and the diameter to be milled. In the sketch of Fig. 646, let "a" equal the circumference and "c" the lead of helix. The greater the lead of the helix for a given diameter, the smaller the helix angle, and the greater the diameter for a given lead, the greater the helix angle. Any change in the diameter of the work or in the lead will make a corresponding change in the helix angle.

$$\frac{\text{Cir.}}{\text{Lead}} = \frac{\text{SO}}{\text{SA}} = \text{Tangent of Helix Angle}$$

$$\frac{\text{Lead}}{10} = \text{Gear Ratio}$$

Lead = Cir. x Cot of Helix Angle

$$\frac{\text{Lead}}{10} = \text{Gear Ratio}$$

Gear Ratio x 10" = Lead Cut

Required lead - lead gears will cut = Error in Lead

Find gearing required to mill the flutes on a 3" diameter cutter (Fig. 645) when the helix angle is 35°8'.

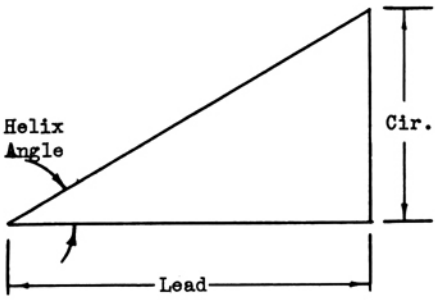


Fig. 647

Lead = Cir. ($\pi \cdot D$) x Cot of Helix Angle

Lead = 3.1416 x 3 x 1.4211 = 13.3935

$$\text{Gear Ratio} = \frac{\text{Lead}}{10} = \frac{13.3935}{10} \times \frac{10000}{10000} = \frac{133935}{100000}$$

100000	133935	1
67870	100000	2
32130	33935	1
30685	32130	17
1445	1805	1
1440	1445	4
5	360	72
	360	
	0	

		1	2	1	17	1	4	72	
0	1	1	3	4	71	75	371	26785	x 5 = 133935
1	0	1	2	3	53	56	277	20000	x 5 = 100000
		1	2	1	17	1	4	72	

$$\frac{75}{56} = \frac{\cancel{3}^{\,1} \times \cancel{48}^{\,16}}{\cancel{64}^{\,4} \times \cancel{56}^{\,7}} = \frac{48}{64} \times \frac{100}{56} = \frac{\text{Driven Gears}}{\text{Driving Gears}}$$

$$\frac{75}{56} \times \frac{10}{1} = \frac{750}{56} = 13.3928 \text{ Lead}$$

13.3935 Required Lead
13.3928 Lead Cut
.0007 Error in Lead

Set table at the helix angle.

Find helix angle and gearing required for a lead of 3.140" on $1\frac{1}{2}$" dia.

$$\frac{\text{Cir.}}{\text{Lead}} = \text{Tangent of Helix Angle}$$

$$\frac{3.1416 \times 1.5}{3.140} = \frac{4.7124}{3.140} = 1.50076$$

1.50076 is the tangent of 56°19'24"

$$\text{Gear Ratio} = \frac{\text{Lead}}{10} = \frac{3.140}{10} = \frac{314}{1000}$$

314	1000	3
290	942	5
24	58	2
20	48	2
4	10	2
4	8	2
0	2	

		3	5	2	2	2	2	
1	0	1	5	11	27	65	157	x 2 = 314
0	1	3	16	35	86	207	500	x 2 = 1000
		3	5	2	2	2	2	

$$\frac{27}{86} = \frac{\cancel{3}^{\,1} \times \cancel{72}^{\,24}}{\cancel{2}^{\,1} \times \cancel{86}^{\,43}} = \frac{24}{64} \times \frac{72}{86} \quad \begin{matrix}\text{Driven Gears} \\ \text{Driving Gears}\end{matrix}$$

$$\frac{27}{\cancel{86}^{\,43}} \times \frac{\cancel{10}^{\,5}}{1} = \frac{135}{43} = 3.13953 \text{ lead cut}$$

Required lead - lead cut = error

3140" - 3.13953" = .00007".

Fig. 648 shows how the indexing head and the rack cutting attachment are set up to mill a helical groove or thread in a worm. The worm shown has a triple thread, 2" pitch diameter, .500 pitch, 1.500 lead, and a helix angle of 76°34'30". The gear ratio and the table setting for this worm are found as follows.

$$\frac{\text{Lead}}{10} = \text{Gear Ratio} \qquad \frac{1.5}{10} \times \frac{10}{10} = \frac{15}{100}$$

$$\frac{15}{100} = \frac{\cancel{3}^{\,1} \times \cancel{40}^{\,24}}{\cancel{64}^{\,8} \times \cancel{100}^{\,25}} = \frac{24}{64} \times \frac{40}{100} \quad \begin{matrix}\text{Driven Gears} \\ \text{Driving Gears}\end{matrix}$$

When using the rack cutting attachment, the cutter is held at a 90° angle with the work and the table is set at the complement of the helix angle.

$$\begin{array}{r} 90^\circ = 89^\circ 59' 60'' \\ \underline{76^\circ 34' 30''} \\ 13^\circ 25' 30'' \end{array}$$

SHORT LEAD MILLING. When very small leads are required, the dividing head worm and worm wheel may be disengaged and the gearing connected directly from the dividing head spindle to the table lead screw. With even gearing, when the dividing head spindle revolves once, the lead screw (which has four threads per inch) makes one revolution and the table is moved a distance equal to the lead, or .250". The rack cutting attachment shown in Fig. 648 is used with this method.

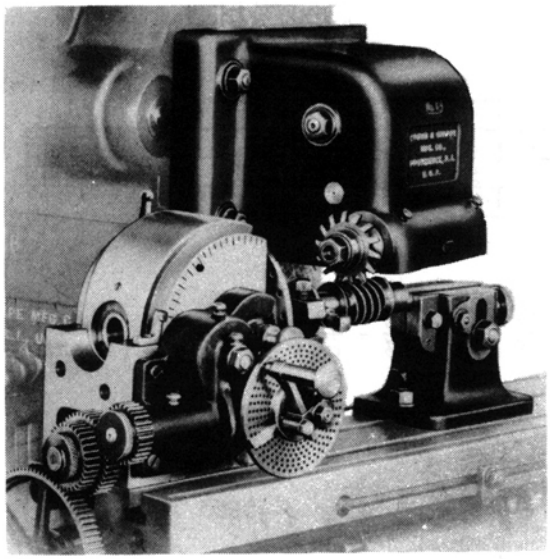


Fig. 648

EXAMPLE. Find the gears to cut a lead of .3492".

$$\frac{\text{Lead}}{.250} = \text{Gear Ratio} \qquad \frac{.3492}{.250} \times \frac{10000}{10000} = \frac{3492}{2500}$$

2500	3492	1
1984	2500	2
516	992	1
476	516	1
40	476	11
36	440	1
4	36	9
	36	

		1	2	1	1	11	1	9	
0	1	1	3	4	7	81	88	873	x 4 = 3492
1	0	1	2	3	5	58	63	625	x 4 = 2500
		1	2	1	1	11	1	9	

$$\frac{88}{63} = \frac{\cancel{8}^{64} \times \cancel{11}^{44}}{\cancel{9} \times \cancel{7}} = \frac{64}{72} \times \frac{44}{28} \quad \begin{array}{l} \text{Driven Gears} \\ \text{Driving Gears} \end{array}$$

$$\frac{88}{63} \times \frac{.250}{1} = .349206 \text{ lead cut}$$

Required lead - lead cut = error

.349206 - .3492 = .000006 error

The regular means of indexing cannot be used in short lead milling. Have the number of teeth in the gear on the spindle some multiple of the number of divisions required. The gears may then be swung out of mesh and

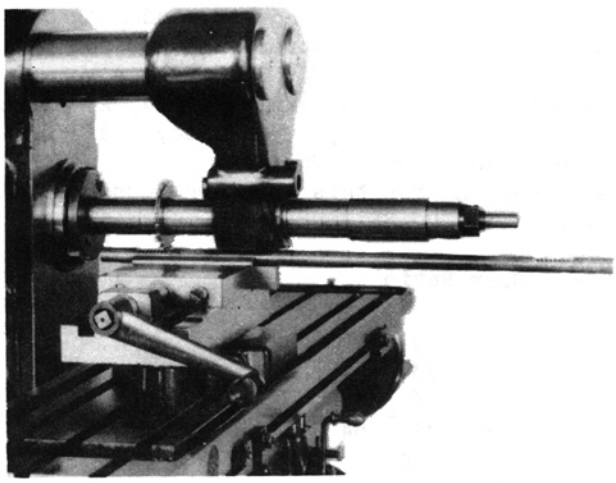


Fig. 649

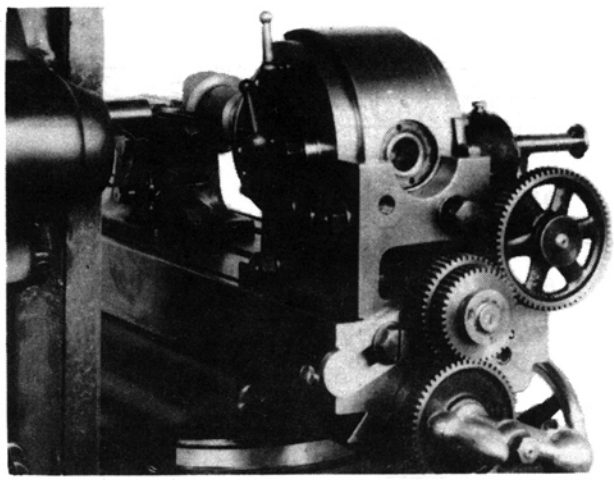


Fig. 650

the gear on the spindle advanced the number of teeth necessary to index the work one division.

Sometimes it is necessary to mill a few teeth on a cylindrical shaft or plunger. If a rack cutting attachment is not available, the work may be done as shown in Fig. 649. The shaft is supported on a parallel and clamped in a vise, and the teeth are indexed by means of the graduated dial on the cross feed screw, the movement being equal to the linear pitch, or 3.1416 divided by the diametral pitch. Before indexing, care should be taken to remove backlash from the screw.

Fig. 650 shows a cylindrical cam being milled with an end mill, producing a helical slot with parallel sides. The dividing head centers are brought to a level with the center of the machine spindle. The table is set at right angles to the spindle and the angle of the helix is obtained by the combination of change gears used. Either right-hand or left-hand helices may be cut in this way by leaving out or adding an extra idler gear. When this method is used for cylindrical cam milling, the gears are calculated and placed the same as for helical milling, as shown in Fig. 645.

The Cam Cutting Attachment in Fig. 651 is used for cutting either face, peripheral, or cylindrical cams from a flat cam former (shown by letter "A"). The cam former is made from a disk about one-half inch thick, on which the required outline is laid out. The disk is machined or filed to the required shape. The table of the machine remains clamped in one position during the cutting, and the necessary rotative and longitudinal movements are contained in the mechanism itself. The rotative movement is obtained by a worm driving a wheel fixed to the spindle of the attachment. The cam former is secured to the face of the worm wheel, and as the wheel revolves, the cam former depresses the sliding rack which in turn drives a pinion geared to another rack in the sliding bed of the attachment. This gives the necessary longitudinal movement on the face of the worm wheel.

Fig. 652 illustrates the cutting of a face cam with the cam cutting attachment. The necessary rotative movement is obtained by hand feed, and the necessary longitudinal movement is produced by the cam former and the

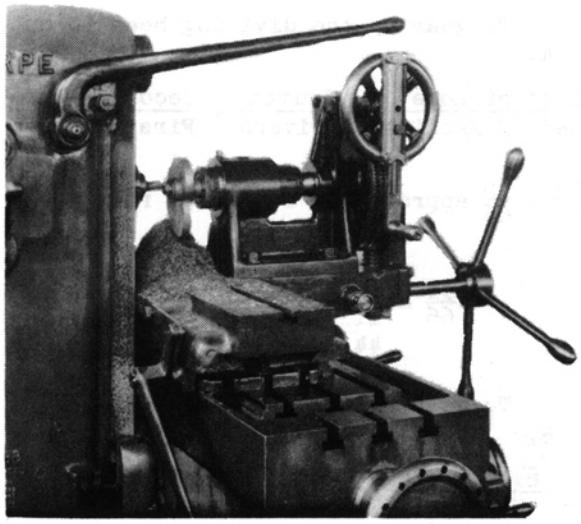


Fig. 652

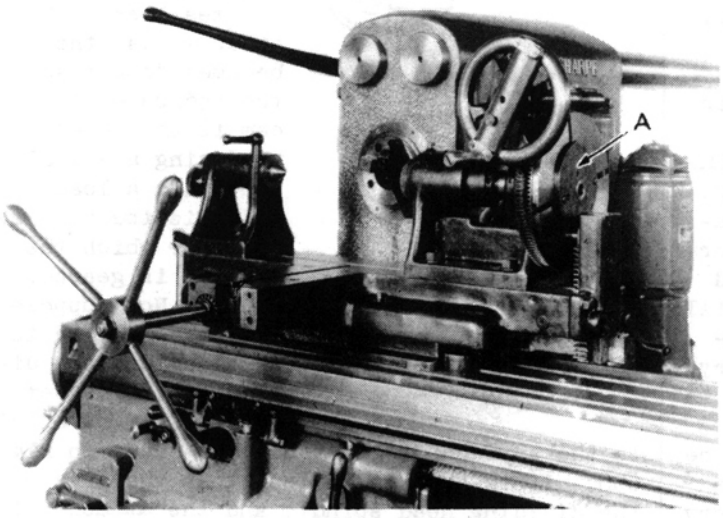


Fig. 651

mechanism of the attachment, as described in the discussion of Fig. 651. A peripheral cam may be milled in the same manner.

The face cam illustrated in Fig. 653 is machined by another method. The work is held in the horizontal plane, in the dividing head, and an end mill is used in the vertical spindle attachment.

In Fig. 653 we have a cam with two lobes (a lobe is a projecting part of a cam wheel), one (A) having a rise of 2.493" in 169°12' and the other (B) having a rise of 2.442" in 104°24'.

The lead in 360° of lobe A

$$= \frac{360^\circ}{169^\circ 12'} \times 2.493 = 5.304"$$

The lead in 360° of lobe B

$$= \frac{360^\circ}{104^\circ 24'} \times 2.442 = 8.421"$$

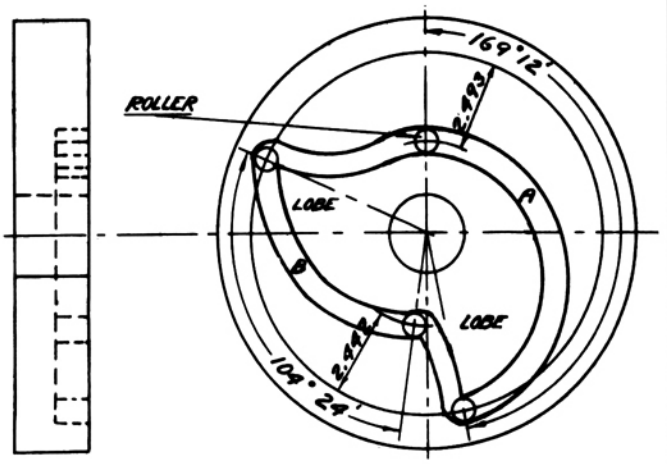


Fig. 653

To gear up the dividing head to cut lobe A,

$$\frac{\text{Lead of Lobe}}{\text{Lead of Machine}} = \frac{\text{Driven}}{\text{Drivers}} = \frac{\text{Second} \times \text{Worm}}{\text{First} \times \text{Screw}}$$

$$\frac{5.304}{10} = \frac{35}{66} \text{ approx. (by continued fractions)}$$

$$\frac{35}{66} = \frac{\overset{28}{\cancel{7}} \times \overset{40}{\cancel{8}}}{11 \times \underset{40}{\cancel{6}}} = \frac{28}{44} \times \frac{40}{48}$$

To gear up the dividing head to cut lobe B,

$$\frac{8.421}{10} = \frac{16}{19} \text{ approx.} = \frac{16 \times 4}{19 \times 4} = \frac{64}{76}$$

(use 2 idlers)

The path of the roller should first be rough drilled. The parts of the cam, other than lobes A and B, can be scribed, drilled, and then milled to the scribed lines.

A method often followed in cutting peripheral cams, especially those for use on automatic screw machines, is that of using the dividing head and a vertical spindle milling attachment. This is illustrated in Fig. 654. The dividing head is geared to the table feed screw, the same as in cutting an ordinary helix, and the cam blank is fastened to the end of the dividing head. An end mill is used in the vertical spindle milling attachment, which is set to mill the periphery of the cam at right angles to its sides. In other words, the axes of the dividing head spindle and attachment spindle must always be parallel to mill cams by this method. The cutting is done by the teeth on the periphery of the end mill. The principle of this method may be explained in the following way.

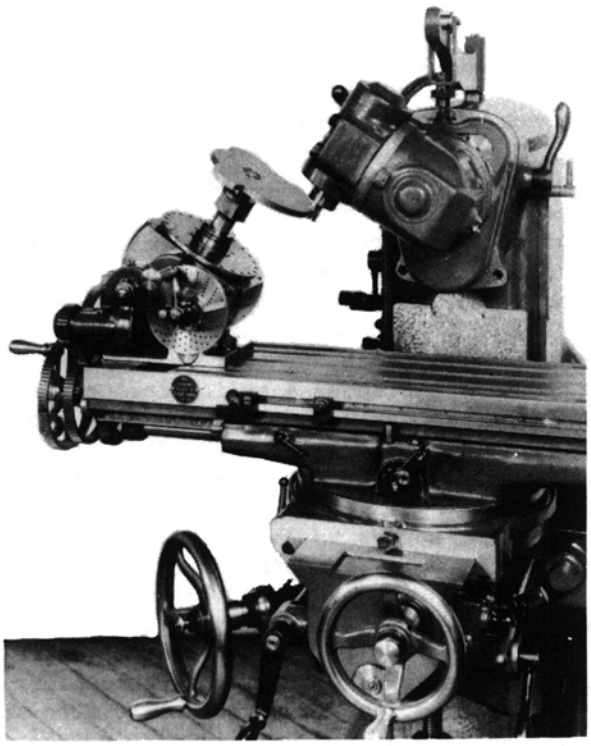


Fig. 654

Suppose the dividing head is elevated to 90°, or at right angles to the surface of the table (see Fig. 655), and is geared for any given lead. It is apparent that as the table advances and the blank is turned, the distance between the axes of the dividing head spindle and the attachment spindle becomes less. In other words, the cut becomes deeper and the radius of the cam is shortened, producing a spiral lobe with a lead which is the same as that for which the machine is geared.

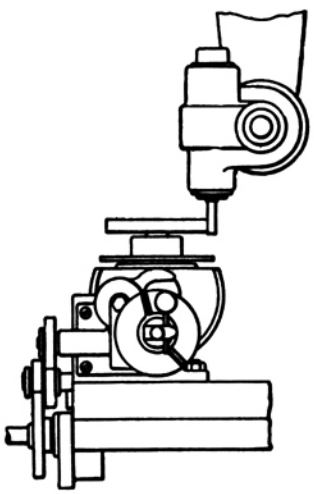


Fig. 655

Now suppose the same gearing is retained and the dividing head is set at zero, or parallel to the surface of the table, as shown in Fig. 656. It is apparent, also, that the axes of the dividing head spindle and the attachment spindle are parallel to each other. Therefore, as the table advances and the blank is

turned, the distance between the axes of the dividing head spindle and the attachment spindle remains the same. As a result, the periphery of the blank, if milled, is concentric, or the lead is zero.

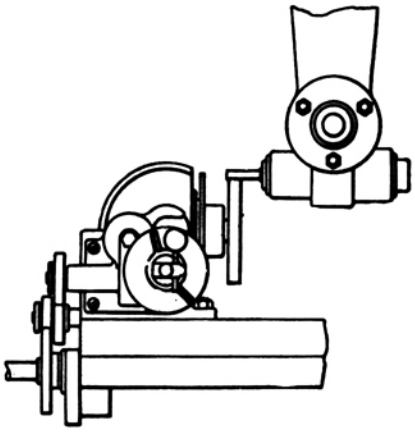


Fig. 656

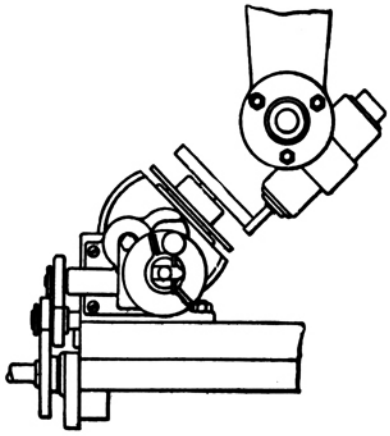


Fig. 657

If, then, the dividing head is elevated to any angle between zero and 90°, as shown in Fig. 657, the amount of lead given to the cam will be between that for which the machine is geared and zero. Hence it is clear that cams with a very large range of different leads can be obtained with one set of change gears, and the problem of milling the lobes of a cam is reduced to a question of finding the angle at which to set the head to obtain any given lead.

To cut the smallest possible lead with the dividing head geared to the lead screw, place a 24-tooth gear on worm, an 86-tooth gear first on stud, a 24-tooth gear second on stud, and a 100-tooth gear on worm. Calculate the lead as follows:

$$\frac{24}{86} \times \frac{24}{100} \times \frac{40}{4} = .66976 \text{ or } .67$$

$\frac{40}{4}$ is number of turns of index crank to one spindle
is number of threads per inch on lead screw

To find the angle to set the dividing head and vertical spindle attachment, divide the lead of the cam by the lead of the machine. The lead of the machine must always be greater than the lead of the cam.

To find the lead of the cam, that is, the theoretical continuous rise in one complete revolution, if the rise in 27° is .127", calculate as follows: 360° ÷ angle in which rise occurs × rise = rise in 360°.

$$\frac{360}{27} \times .127 = 1.693 \text{ rise in } 360°$$

To calculate the leads, gears, and angles to incline the dividing head and vertical spindle attachment for a cam having a .470" rise in 85°, .750" rise in 75°, and .358" rise in 58° (see Fig. 658), proceed as follows:

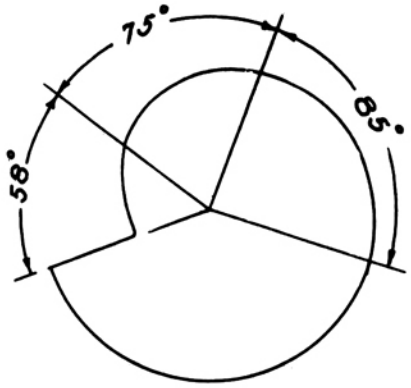


Fig. 658

$$.470" \text{ rise in } 85° = \frac{360}{85} \times .470$$

$$= 1.99$$

$$.750" \text{ rise in } 75° = \frac{360}{75} \times .750 = 3.600$$

$$.358" \text{ rise in } 58° = \frac{360}{58} \times .358 = 2.222$$

The machine must be geared with a greater lead than that of the cam having the greatest lead. As an example, use the lead 3.657" and gear the machine as follows:

$\frac{3.657}{10.000}$ by continued fractions equals

$$\text{approximately } \frac{64}{175} = \frac{\cancel{32}}{\cancel{7}} \times \frac{64}{\cancel{25}} \quad \begin{matrix} 32 & 64 \\ 56 & 100 \end{matrix}$$

Place the 32-tooth gear on the worm, the 56-tooth gear the first on the stud, the 64-tooth gear the second on the stud, and place the 100-tooth gear on the lead screw.

The sine of the angle at which to incline the dividing head is found by dividing the lead of the cam by the lead of the machine.

$$\text{Lead of cam having .470" rise in } 85° = \frac{1.99}{3.657}$$

$$= .54415 \text{ sine} = 32°58'$$

$$\text{Lead of cam having .750" rise in } 75° = \frac{3.600}{3.657}$$

$$= .9844 \text{ sine} = 79°52'$$

Lead of cam having .358" rise in 58° = $\frac{2.222}{3.657}$

= .6076 sine = 37°25'

Note in the above work that the machine is geared the same for all leads, and that the dividing head and vertical spindle attachment are inclined at different angles to mill the different leads.

A cylindrical cam is milled and the gears calculated in the same manner as a helical groove is milled, an end mill being used instead of a milling cutter.

All tool rooms have, or can obtain, a chart showing the many different leads and gears used to cut these leads.

CUTTER SPEEDS

The cutting speed of a cutter is the rate at which it engages the work (usually expressed in surface feet per minute). No definite rule can be made for the speed of a milling cutter, because too many factors, such as the depth of cut, amount of feed, material cut, type of job, condition of machine and cutter, and finish required, must be considered. With H.S.S. cutters, the following feeds are usually satisfactory:

High Speed Steel (H.S.S.)	- 40 -	50	F.P.M.
Chrome Non-Shrink (C.N.S.)	- 40 -	50	"
Tool Steel (R RR)	- 60 -	80	"
Machine Steel (M.S.)	- 80 -	100	F.P.M.
Cast Iron (C.I.)	- 80 -	100	"
Brass	- 150 -	200	"

A cutter should never be run at a speed which would cause excessive heat and dull or burn the cutting edge.

The following formulas may be used in figuring the speed of a milling cutter.

(a) $$R.P.M. = \frac{\text{Cutting Speed} \times 12}{3.1416 \times \text{dia.}}$$

Example. What is the speed of the spindle for a $1\frac{1}{4}$" cutter running 40 F.P.M.?

$$R.P.M. = \frac{40 \times 12}{3.1416 \times 1.25} = 122+$$

(b) $$\text{Cutting Speed} = \frac{3.1416 \times \text{dia.} \times R.P.M.}{12}$$

Example. Find the cutting speed of a $1\frac{1}{2}$" end mill running 382 R.P.M.

$$\text{Cutting Speed} = \frac{3.1416 \times 1.5 \times 382}{12} = 150+$$

The following table may be used to find directly the R.P.M. of cutters of different diameters (for the more common surface speeds):

TABLE OF CUTTING SPEEDS

Feet per Minute	40	45	50	55	60	65	70	75	80	90	100	110	120	130	140	150
Diam., Inches	REVOLUTIONS PER MINUTE															
1/4	611	688	764	840	917	993	1070	1146	1222	1375	1528	1681	1833	1986	2139	2292
5/16	489	550	611	672	733	794	856	917	978	1100	1222	1345	1467	1589	1711	1833
3/8	407	458	509	560	611	662	713	764	815	917	1019	1120	1222	1324	1426	1528
7/16	349	393	437	480	524	568	611	655	698	786	873	960	1048	1135	1222	1310
1/2	306	344	382	420	458	497	535	573	611	688	764	840	917	993	1070	1146
5/8	244	275	306	336	367	397	428	458	489	550	611	672	733	794	856	917
3/4	204	229	255	280	306	331	357	382	407	458	509	560	611	662	713	764
7/8	175	196	218	240	262	284	306	327	349	393	437	480	524	568	611	655
1	153	172	191	210	229	248	267	287	306	344	382	420	458	497	535	573
1-1/8	136	153	170	187	204	221	238	255	272	306	340	373	407	441	475	509
1-1/4	122	138	153	168	183	199	214	229	244	275	306	336	367	397	428	458
1-3/8	111	125	139	153	167	181	194	208	222	250	278	306	333	361	389	417
1-1/2	102	115	127	140	153	166	178	191	204	229	255	280	306	331	357	382
1-5/8	94.0	106	118	129	141	153	165	176	188	212	235	259	282	306	329	353
1-3/4	87.3	98.2	109	120	131	142	153	164	175	196	218	240	262	284	306	327
1-7/8	81.5	91.7	102	112	122	132	143	153	163	183	204	224	244	265	285	306
2	76.4	85.9	95.5	105	115	124	134	143	153	172	191	210	229	248	267	287
2-1/4	67.9	76.4	84.9	93.4	102	110	119	127	136	153	170	187	204	221	238	255
2-1/2	61.1	68.8	76.4	84.0	91.7	99.3	107	115	122	138	153	168	183	199	214	229
2-3/4	55.6	62.5	69.5	76.4	83.3	90.3	97.2	104	111	125	139	153	167	181	194	208
3	50.9	57.3	63.7	70.0	76.4	82.8	89.1	95.5	102	115	127	140	153	166	178	191
3-1/4	47.0	52.9	58.8	64.6	70.5	76.4	82.3	88.2	94.0	106	118	129	141	153	165	176
3-1/2	43.7	49.1	54.6	60.0	65.5	70.9	76.4	81.9	87.3	98.2	109	120	131	142	153	164
3-3/4	40.7	45.8	50.9	56.0	61.1	66.2	71.3	76.4	81.5	91.7	102	112	122	132	143	153
4	38.2	43.0	47.7	52.5	57.3	62.1	66.8	71.6	76.4	85.9	95.5	105	115	124	134	143
4-1/2	34.0	38.2	42.4	46.7	50.9	55.2	59.4	63.6	67.9	76.4	84.9	93.4	102	110	119	127
5	30.6	34.4	38.2	42.0	45.8	49.7	53.5	57.3	61.1	68.8	76.4	84.0	91.7	99.3	107	115
5-1/2	27.8	31.3	34.7	38.2	41.7	45.1	48.6	52.1	55.6	62.5	69.5	76.4	83.3	90.3	97.2	104
6	25.5	28.6	31.8	35.0	38.2	41.4	44.6	47.8	50.9	57.3	63.7	70.0	76.4	82.8	89.1	95.5

Chapter 22

GAGES AND GAGE BLOCKS

1. What is a gage?

A. A gage is a device for determining whether or not one or more of the dimensions of a manufactured part are within specified limits. A master gage is one whose gaging dimensions represent as exactly as possible the physical dimensions of the component. An inspection gage is a gage for the use of the manufacturer or purchaser in accepting the product, and a working gage is one used by the manufacturer to check the work as it is produced.

2. Name several types of gages.

A. Gages are divided into the following types according to the purposes for which they are used: ring gage, plug gage, receiving gage, indicating gage, snap gage, and caliper gage.

Fig. 659. Ring Gage

3. What is a ring gage?

A. A ring gage, Fig. 659, is one whose inside measuring surfaces are circular in form. The measuring surfaces may be cylindrical or conical.

4. What is a plug gage?

A. A plug gage is one whose outside measuring surfaces are arranged to verify the specified uniformity of holes. A plug gage may be straight or tapered and of any cross-sectional shape. Figs. 660, 661 and 662 show different types of plug gages.

Fig. 660. Cylindrical Plug Gage

Fig. 661. Spline Plug Gage

5. What is a snap gage, and what is a caliper gage?

A. A snap gage, Fig. 663, is a fixed gage arranged with inside measuring surfaces for calipering diameters, lengths, thicknesses, etc., and a caliper gage, Fig. 664, is one which, for internal members, is similar to a snap page, and for external members, is similar to a plug gage.

Fig. 662. Taper Plug Gage

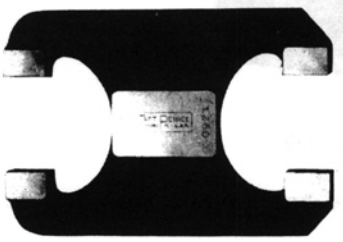


Fig. 663

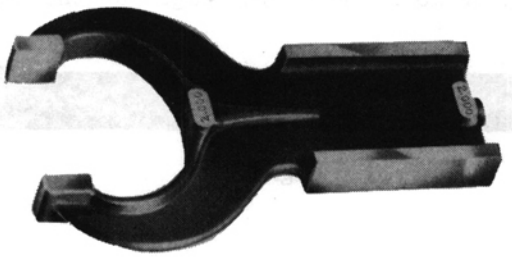


Fig. 664

Fig. 665

6. What is an indicating gage?

A. An indicating gage, Fig. 666, is one that exhibits visually the variations in the uniformity of dimensions or contour, the amount of the variation being indicated by lever on graduated scale, dial, flush pin, plunger, gages, etc.

7. What is a receiving gage?

A. A receiving gage, Fig. 665, is one whose inside measuring surfaces are arranged to verify the specified uniformity of size and contour of manufactured material.

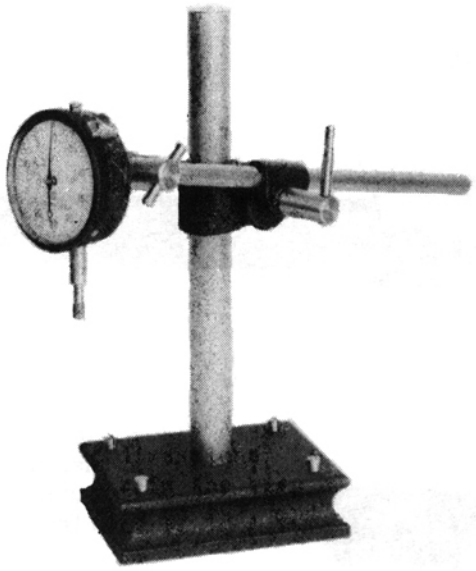


Fig. 666

8. Explain what a sine bar is and tell how it is used.

A. The sine bar is used either for measuring angles accurately or for locating work to a given angle within very close limits. It consists of a bar in which 2 hardened and grouped plugs of the same diameter are set. The center distance between the plugs is usually 5 or 10 inches. The edges of the bar must be parallel with the line of the plug centers. The sine bar is always used in conjunction with a true surface, such as a surface plate from which measurements are taken. Fig. 667 shows a 10" straight sine bar and Fig. 669 shows how it is used. In Fig. 668, the difference in the height of the measuring plugs, A-B, divided by the length of the sine bar is equal to the sine of the required angle.

Fig. 667

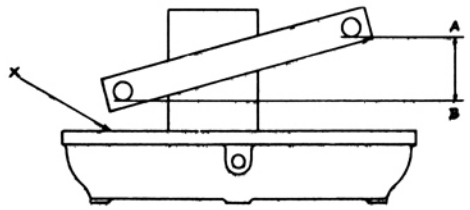


Fig. 668

For example, in setting a 5" sine bar, as shown in Fig. 668, distance A-B was determined by means of a Vernier Height Gage to be 2.68525". This number, divided by 5, (because the 5" sine bar was used) gave a result of 0.53705", the sine of the angle required. A table of natural sines will show the value of this angle to be 32°29'. If a 10" sine bar were used, distance A-B would have been 5.3705", or 10 times the natural sine of an angle of 32°29'. An error in setting the 10" sine bar would cause but half the inaccuracy in measurement of the angle that would be caused by the same error in setting the 5" bar. In setting up for a given angle, the method outlined above is reversed. By referring to a table of natural sines, the sine of an angle of 32°29' is found to be 0.53705". Then by setting the 5" sine bar in a manner such that the perpendicular distance A-B is five times the natural sine, or 2.68525", the correct angle is at once established between the edge of the sine bar and the surface plate on which it is mounted.

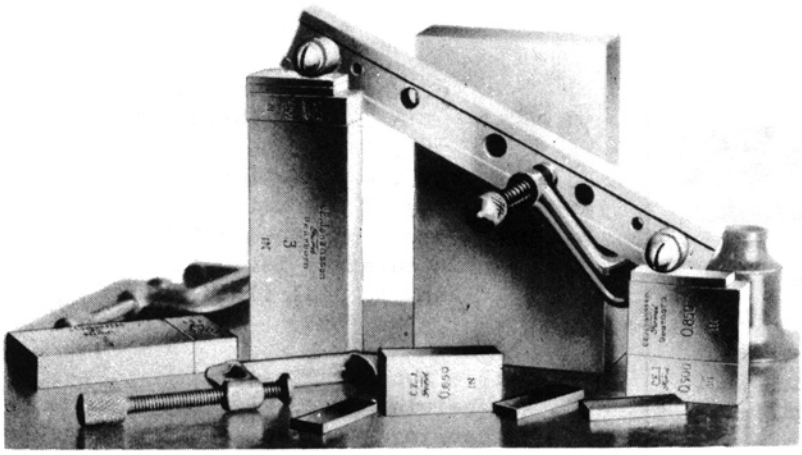


Fig. 669. Using Straight Sine Bar

9. Tell what the amplifying comparator, Fig. 671, is and explain its use.

A. The amplifying comparator is a precision measuring instrument which amplifies any error in the work 10 times.

A combination of Jo-blocks, or a gage of predetermined size is placed under

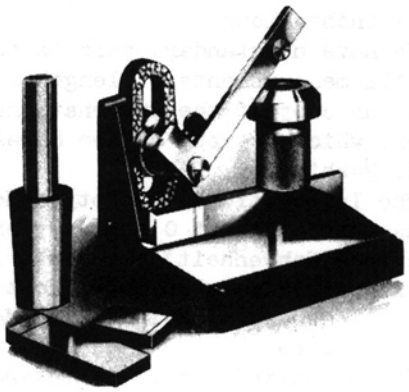


Fig. 670. Using Sine Bar Fixture

the indicating point and the dial indicator hand set at zero. The Jo-blocks or gage is then removed and the job to be checked is placed under the point. Any deviation shown on the dial graduations indicates ten times the amount of error between the work and the pre-determined size. Gages should be checked for accuracy at 68°F.

10. In fitting a shaft to a hole, which is usually made standard, the hole or shaft?
A. Usually the hole is made standard and any allowance for fit is made on the shaft.

When a micrometer has been used with the spindle locked in one place for some time checking hardened work such as reamers, cutters, or similar jobs, the end of the spindle and the anvil will be worn slightly concave, Fig. 673. The spindle and the anvil should be checked occasionally with a straight edge, Fig. 674, to see whether they are worn or not. If these surfaces are worn, they should be lapped flat and parallel with

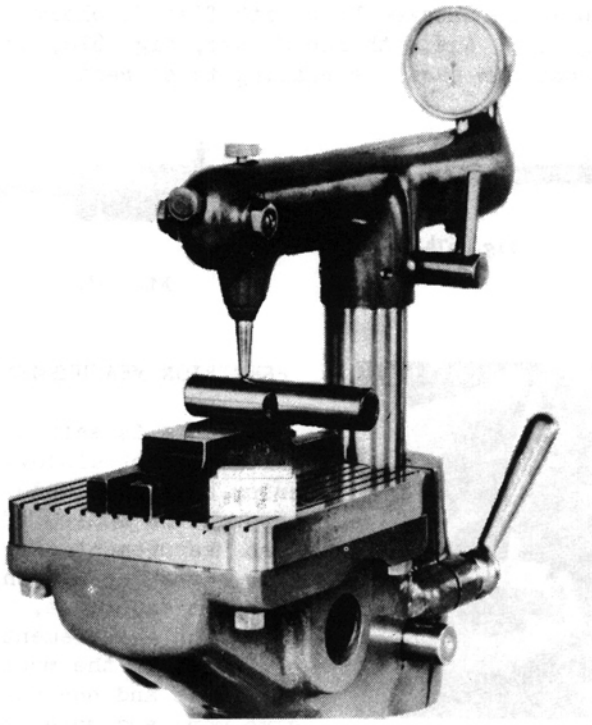


Fig. 671

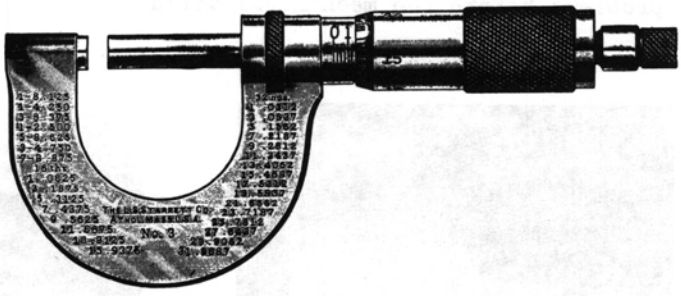
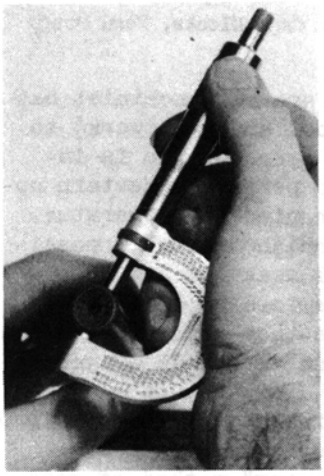
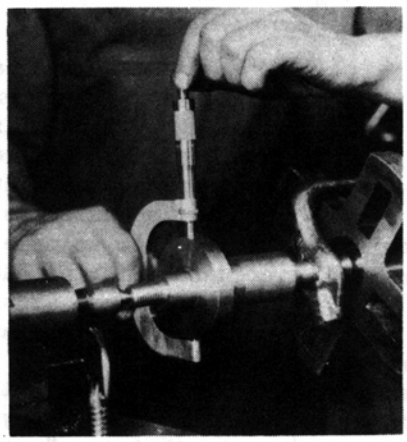


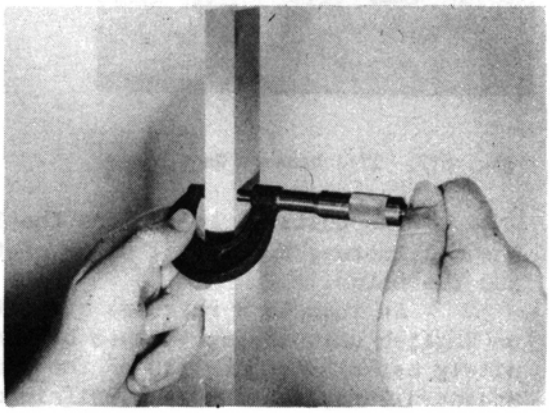
Fig. 673



(a)



(b)



(c)

Fig. 672. Suggestions for Holding a Micrometer

each other. When adjustments are made they should be checked both with flat Jo-block, Fig. 675, and with round jaws, Fig. 676, to be certain that the reading is correct.

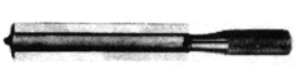


Fig. 674

Fig. 676

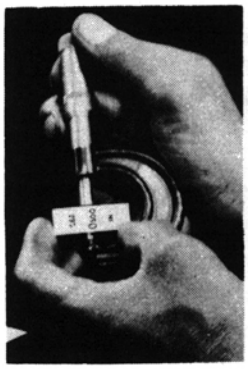


Fig. 675

PRECISION MEASUREMENT

It is said that all important discoveries have been made through the medium of fine measurement of Time, Mass, or Length, and of these three, precision measurement of length is the most difficult and has become more and more important until today it is one of the greatest problems before the mechanical world.

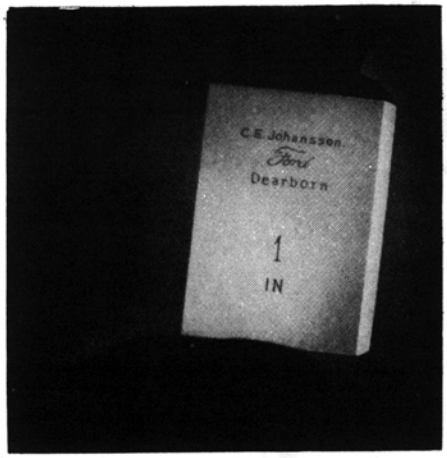


Fig. 677. 1" Johansson Gage Block

The units of length are the Inch and the Millimeter, fractional parts of the yard and the meter.

An inch is a relative quantity and is an English unit of length that has been definitely established at 1/36 of the "Imperial Standard Yard," which Great Britain defines as the distance at 62 degrees Fahrenheit between the central traverse lines in two gold plugs in a bronze bar one inch square and thirty-six inches long.

We have no standard yard in the United States. All measurements of length are referred to our copy of the International Prototype Meter, which is kept at the Bureau of Standards, Washington, D.C.

The International Prototype Meter is defined as the length at 0 degrees centigrade (or 32 degrees Fahrenheit) between two lines on a platinum-iridium alloy bar, kept in the International Bureau of Weights and Measures near Paris, France.

In the United States, the yard has been defined by Executive Order, dated April 15, 1893, as 3600/3937 of a meter, or expressed as a decimal 0.9144 meters.

An inch in the United States is defined as 1/36 of a yard or 2.54 centimeters.

Johansson Gage Blocks are the standard of precision measurement for the world. They have no counterpart in either science or industry. They measure accurately in millionths of an inch, an accomplishment considered impossible before their introduction.

A few years ago, the mechanics who worked in machine shops were "all around" machinists; that is, they were proficient in the operation of all the machine equipment as well as being able to perform bench and assembly operations.

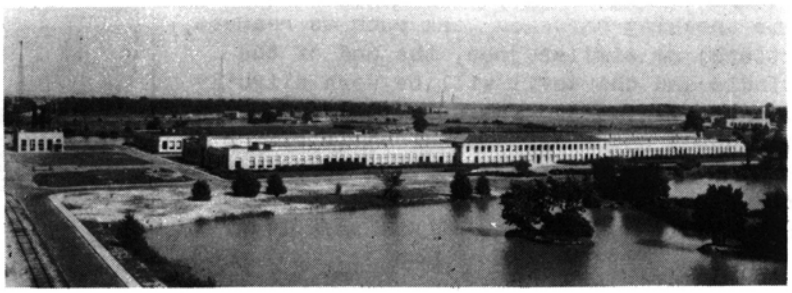


Fig. 678. The "Home" of Johansson Gage Blocks, Ford Motor Company, Dearborn, Michigan.

Today the "all around" machinist has given way (except on tool and gage work) to the highly specialized operator who is instructed and trained to perform a certain operation on a particular piece of apparatus. The tools and gages furnished make it possible for him to make all measurements in accordance with pre-determined specifications which are more accurate than were thought possible by the "all around" mechanic of yesteryear.

The gradually increasing quantity production of recent years is on such a huge scale that the "cut and try" method of measuring has been discarded, even by the machine shop that builds only one complete apparatus.

The ideal measuring tool for making possible precision and interchangeable manufacture must meet the following requirements:

First--Must give external and internal measurements in one ten-thousandth part of an inch.

Second--Must be of such design to be used directly on the work, to eliminate the possibility of error in transferring accurate measurements.

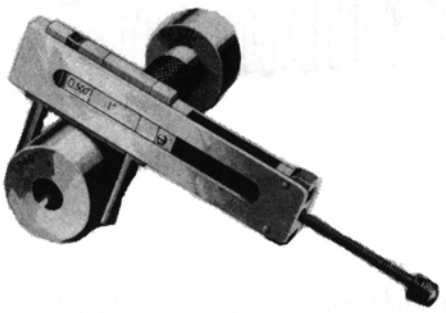


Fig. 679. Measuring OD with Accessory Set

Third--Must be positive, and dependable in a sense, foolproof.

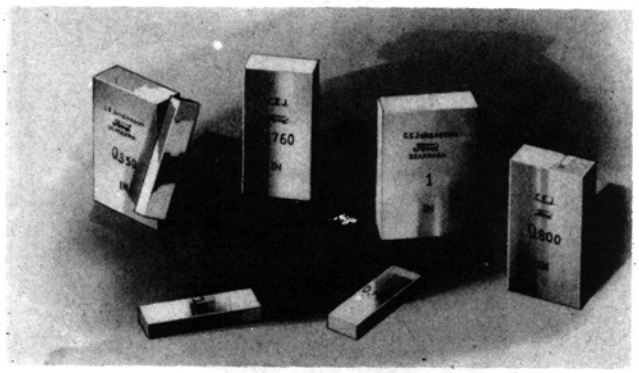


Fig. 683

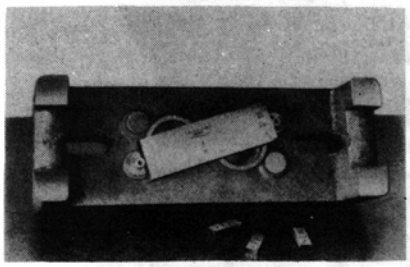


Fig. 680. Checking Distance Between Test Plugs

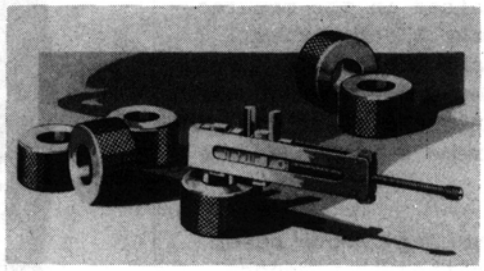


Fig. 681. Measuring ID with Accessory Set

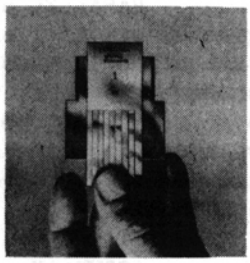


Fig. 682. Measuring with Accuracy.

Fourth--Must give the same result, any number of times, in the hands of different mechanics.

Fifth--Must be self-checking, so error due to wear, accident, or abuse may be readily discovered.

Sixth--Must be seasoned and stabilized to reduce to a minimum errors in accuracy due to the change that takes place in metals.

Seventh--Must have an established reputation for accuracy that is accepted by the manufacturer and customer.

The Johansson Gage Blocks (Fig. 683) are rectangular pieces of tool steel, approximately 3/8 inches by 1-3/8 inches by specified size, hardened, ground, stabilized, and finished to an accuracy within a few millionths part of an inch from specified size. They embody in their commercial manufacture the solving of four universally recognized metallurgical and mechanical problems, namely: Flat Surfaces in Steel; Parallel Surfaces in Steel; Accuracy as to Dimension in Steel; and Effective Heat Treatment and Seasoning of Steel.

FLAT SURFACES IN STEEL. It is considered one of the most remarkable achievements in mechanics to make a flat surface in steel, and, by the Johansson methods, a flat surface with an extremely high finish having the appearance of burnished silver is produced, which approaches nearer the perfect plane than any other surface produced by the hand of man. These flat-lapped surfaces when thoroughly cleaned and slid one on the other with a slight inward pressure, will take hold as though magnetized. They have been known to sustain a weight of 200 pounds on a direct pull (Fig. 684), although the contacting surfaces are less than one-half square inch. Scientists have offered "atmospheric pressure,"

"molecular attraction," and "a minute film of oil on the lapped surfaces" as an explanation of this phenomena. Possibly it is a combination of all three.

Fig. 684

PARALLEL SURFACES IN STEEL. The degree of parallelism attained in the manufacture of the Johansson Gage Blocks is well shown by the fact that any block in a given combination may be turned end for end at will without affecting either the size or the parallelism of the two extreme surfaces of the combination.

ACCURACY. The making of one steel surface parallel with another is a recognized problem, but to make the parallel surface a pre-determined distance from another surface and with an accuracy in millionths of an inch is a most remarkable achievement. That this has been accomplished is proven by the way in which an equivalent combination of Johansson Gage Blocks check against one solid block. See Fig. 685.

EFFECTIVE SEASONING. Johansson Gage Blocks are so seasoned by Johansson methods that internal stresses and strains are relieved. The molecules of the steel may be said to be at rest and because of this, the usual warping or growing is checked.

GAGING SYSTEM. Set No. 1 will make .0001 inch sizes from .200 inch and .001 inch sizes from .100 inch up to 12.000 inches.

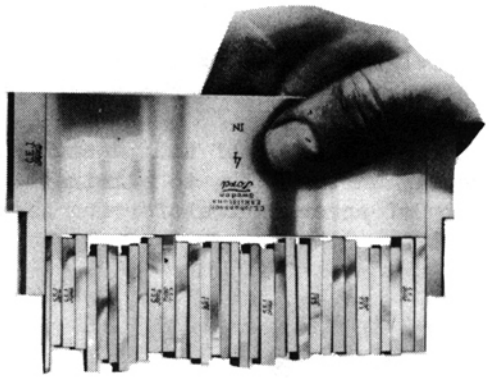


Fig. 685

A full set consists of eighty-one blocks (Fig. 686), which have surfaces flat and parallel within .000008 of an inch. This set is made up of four series:

Fig. 686

Sizes Composing a Set of 81 Blocks

FIRST SERIES

.1001″ .1002″ .1003″ .1004″ .1005″ .1006″ .1007″ .1008″ .1009″

SECOND SERIES

.101″	.102″	.103″	.104″	.105″	.106″	.107″	.108″	.109″	.110″
.111″	.112″	.113″	.114″	.115″	.116″	.117″	.118″	.119″	.120″
.121″	.122″	.123″	.124″	.125″	.126″	.127″	.128″	.129″	.130″
.131″	.132″	.133″	.134″	.135″	.136″	.137″	.138″	.139″	.140″
.141″	.142″	.143″	.144″	.145″	.146″	.147″	.148″	.149″	

THIRD SERIES

.050″

.100″	.200″	.300″	.400″	.500″	.600″	.700″	.800″	.900″
.150″	.250″	.350″	.450″	.550″	.650″	.750″	.850″	.950″

FOURTH SERIES

1.000″ 2.000″ 3.000″ 4.000″

1. The first series consists of nine blocks, ranging in size from 0.1001 of an inch to 0.1009 of an inch by steps of 0.0001 of an inch.
2. The second series consists of forty-nine blocks, ranging in size, by 0.001 of an inch, from 0.101 of an inch to 0.149 of an inch.
3. The third series consists of nineteen blocks, ranging in size, by 0.050 of an inch, from 0.050 of an inch to 0.950 of an inch.
4. The fourth series consists of four blocks, 1, 2, 3, and 4 inches in size.

The blocks of the third series can be combined with those of the fourth series to give any multiple of 0.050 between 0.050 and 10 inches. The second series is used to obtain dimensions varying by thousandths, and the first series to obtain dimensions varying by ten-thousandths.

Johansson Gage Blocks are made in three qualities--B, A, and AA. At a temperature of 68°F, these blocks have the following accuracy:

Working Set (B quality)	= 0.000008 inch
Inspection Set (A quality)	= 0.000004 "
Laboratory Set (AA quality)	= 0.000002 "

The extreme accuracy of these blocks can be indicated by the following comparisons. A human hair is approximately three thousandths (.003) of an inch thick. The most accurate work in the mechanical field is that of toolmakers, who work to an accuracy of one ten-thousandth of an inch, which is thirty times finer than a human hair. To carry the illustration still further, light waves are approximately sixteen millionths of an inch long, which is 250 times finer than a human hair and 6¼ times finer than the accuracy used by a toolmaker. The accuracy of AA quality Johansson Gage Blocks, however, is two millionths (.000002) of an inch. This is

1500 times finer than a human hair;
50 times finer than a toolmaker works;
8 times finer than the length of a light wave.

JOHANSSON BLOCKS

ASSIGNMENT. To build a combination of Johansson blocks of any required size, within the range of the set.

HOW IT IS DONE

(A) TO FIND THE SIZE OF THE BLOCKS TO BE USED.

1. Acquaint yourself with the size of the blocks in the set.
2. Begin with the right-hand figure of the specified size.
3. Continue working from the right to the left.
4. Build the combination with the fewest possible number of blocks.

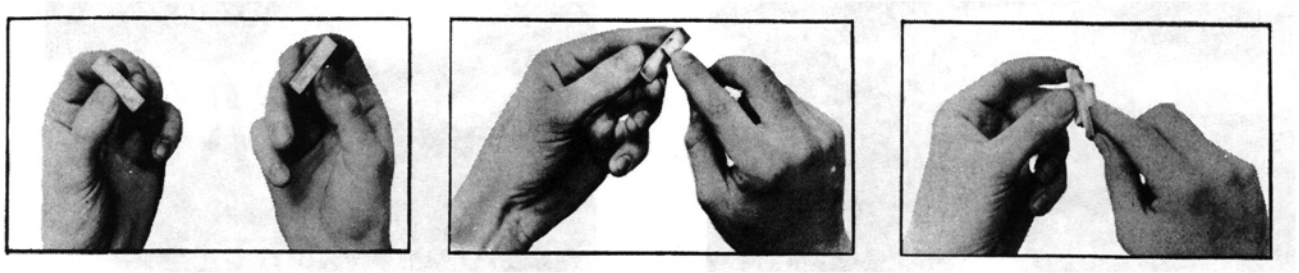
EXAMPLE

1.2721	*1.2721*	*1.2721*	*1.2721*	*1.2721*
.1001	.1009	.1008	.1006	.1007
.149	.1002	.1003	.1005	.1004
.123	.147	.139	.138	.141
.900	.124	.132	.133	.130
1.2721	.800	.100	.500	.600
	1.2721	.700	.300	.200
		1.2721	1.2721	1.2721

It is possible to make many combinations to obtain any required size.

(B) TO BUILD THE COMBINATION

1. Select from the set the first 2 blocks of the combination (Fig. 687(a)).
2. Wipe each of the contacting surfaces of the blocks on the palm of the hand, on the wrist, or on a piece of chamoise, and then place the contacting surfaces together (Fig. 687(b)).
3. With a slight inward pressure, slide one block on the other (Fig. 687(c)). If the contacting surfaces are clean they will cling together as though they were magnetized.
4. Continue in this manner until the required combination is completed.



(a) (b) (c)

Fig. 687

YOUR JOB. Make as many combinations of blocks as possible to form the following sizes: _____; _____; _____.
Do not use the same block for more than one combination.

APPLICATIONS OF JOHANSSON BLOCKS

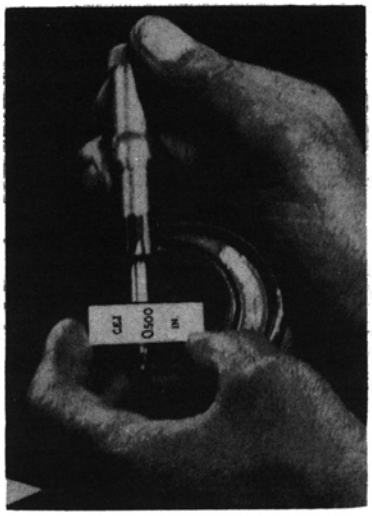


Fig. 688. Checking Micrometer

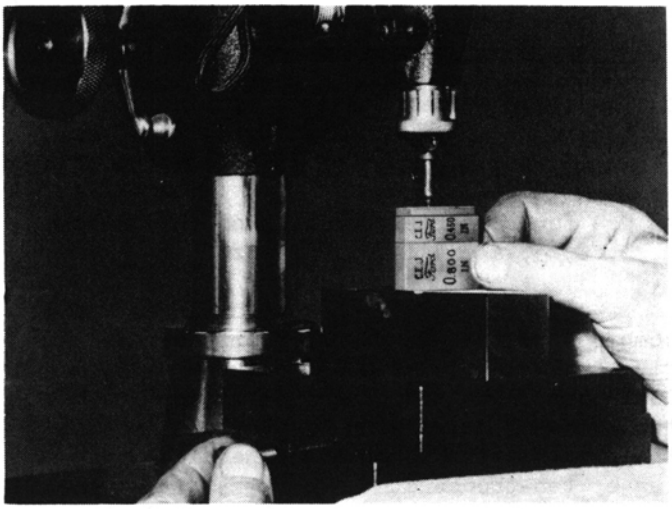


Fig. 689. Setting Visual Gage

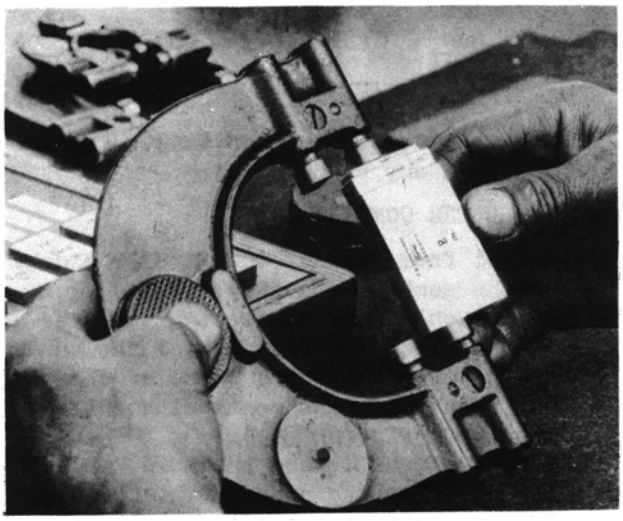


Fig. 690. Checking Snap Gage

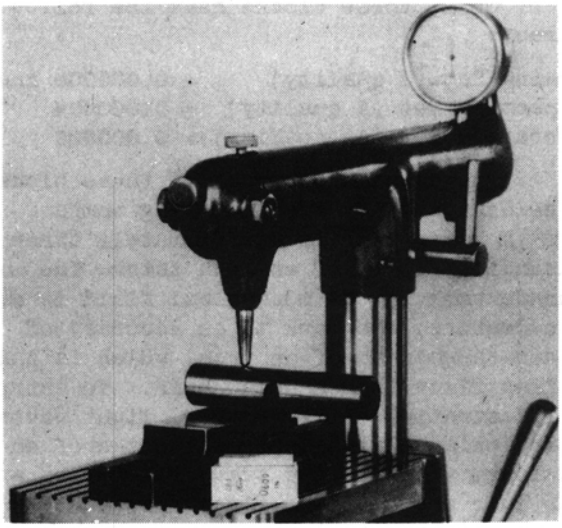


Fig. 691. Using Amplifying Gage Set With Jo-Blocks

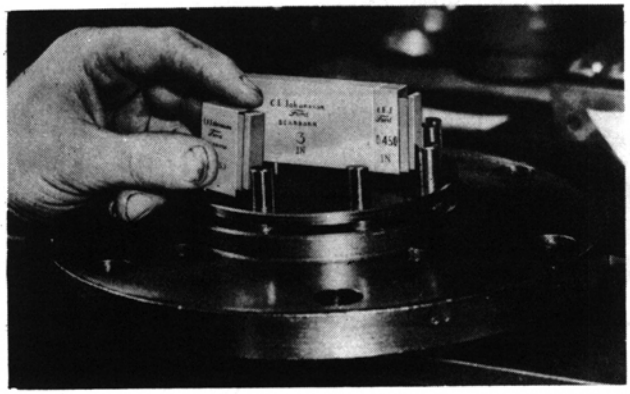


Fig. 692. Checking Spacing of Holes in Relation Gage

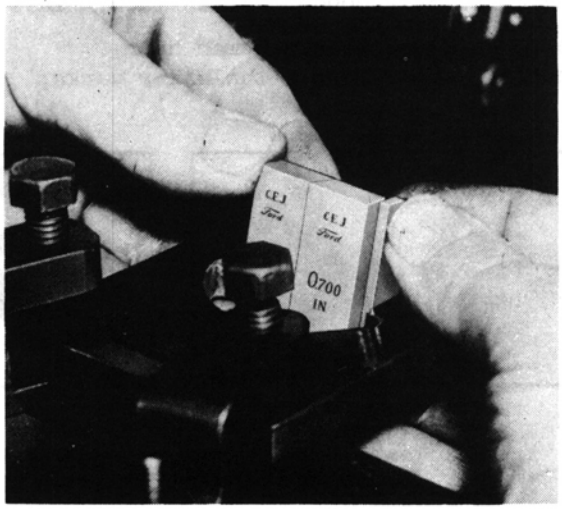


Fig. 693. Measuring Snap Gage on a Grinder

APPLICATION OF JOHANSSON BLOCKS

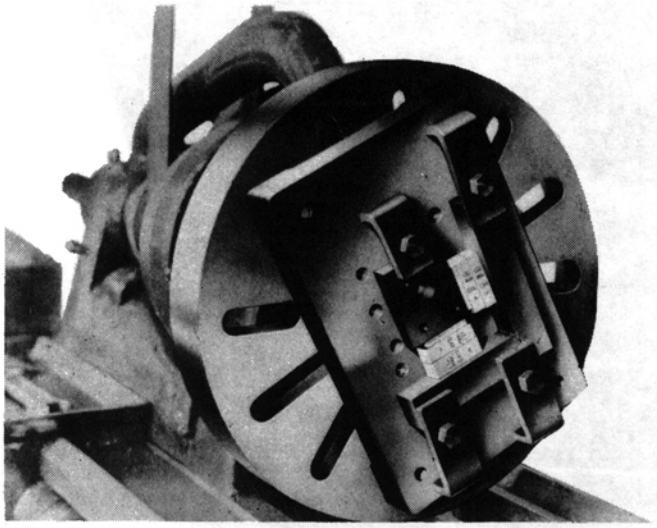


Fig. 694. Gage Blocks Used to Establish Locations

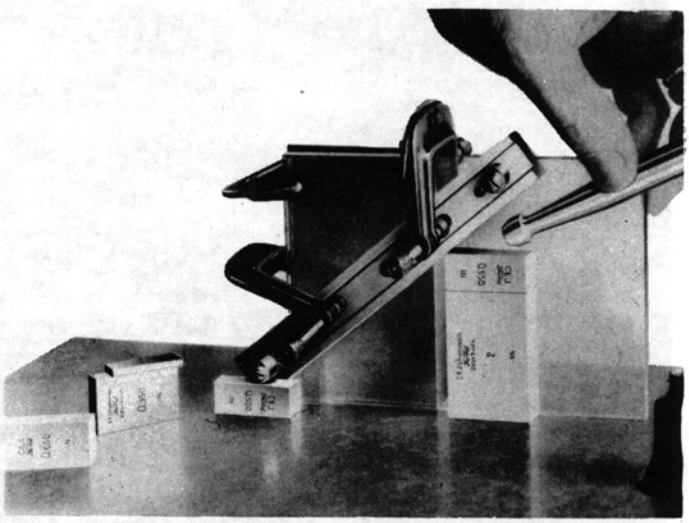
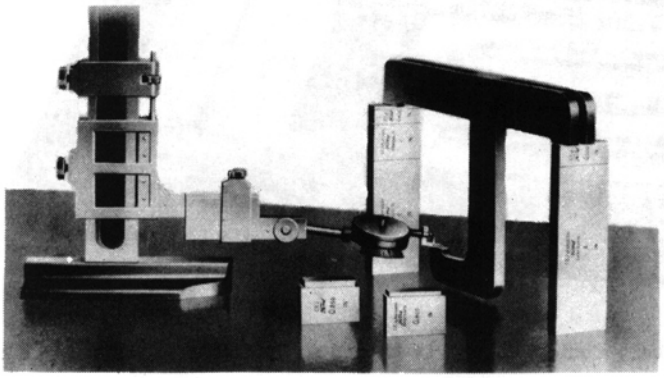
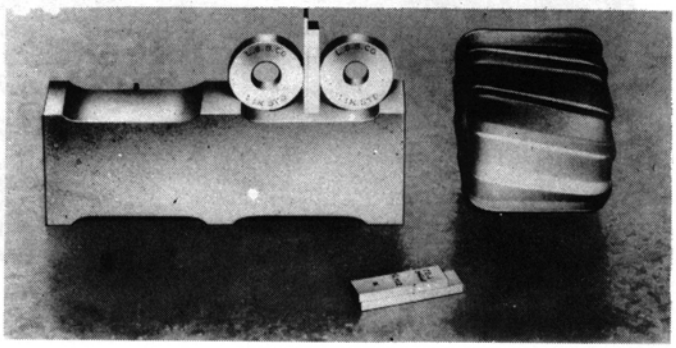


Fig. 695. Checking Taper on Plug Gage

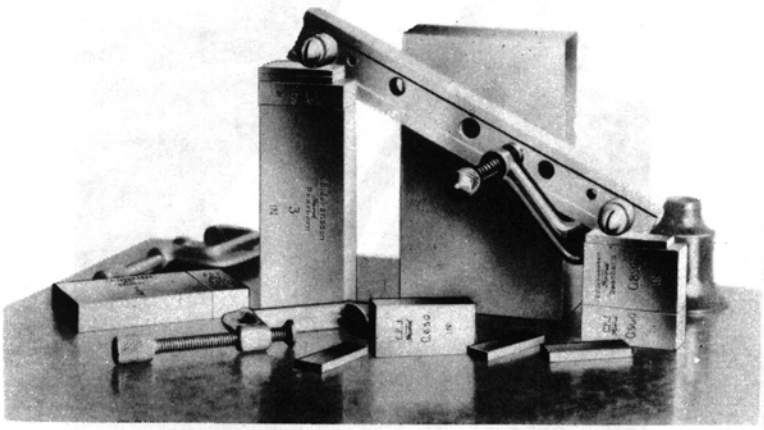


(a)

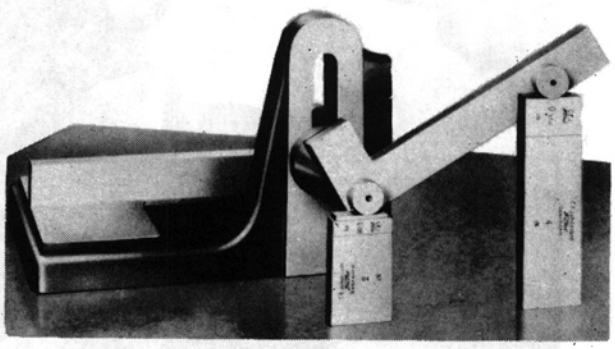


(b)

Fig. 696. (a) Checking Gage with Gage Blocks and Indicator, (b) Checking Location of Radii



(a)



(b)

Fig. 697. Setting Sine Bars with Gage Blocks

APPLICATION OF BLOCKS AND ACCESSORIES

Fig. 698. Accessory Set Containing Jaws, Scriber, Center Point, Tram Points, Straight Edge, Adjustable Holder, and Foot Block.

Fig. 699. Checking Inside Diameter of a Ring Gage

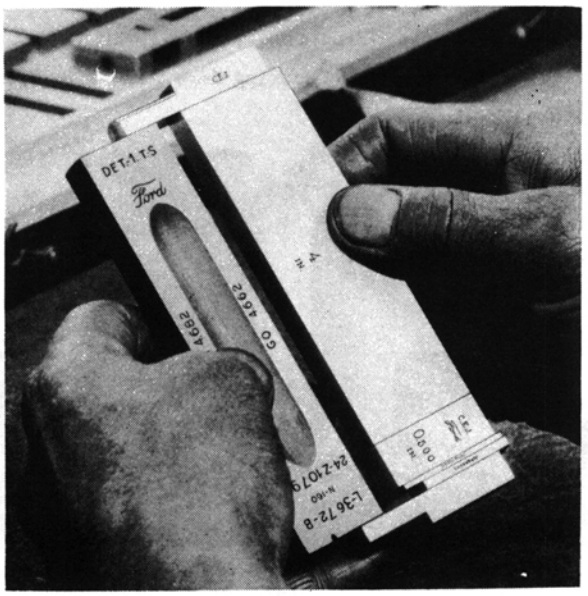


Fig. 700. Checking "GO" Size of Special Length Gage

APPLICATION OF BLOCKS AND ACCESSORIES

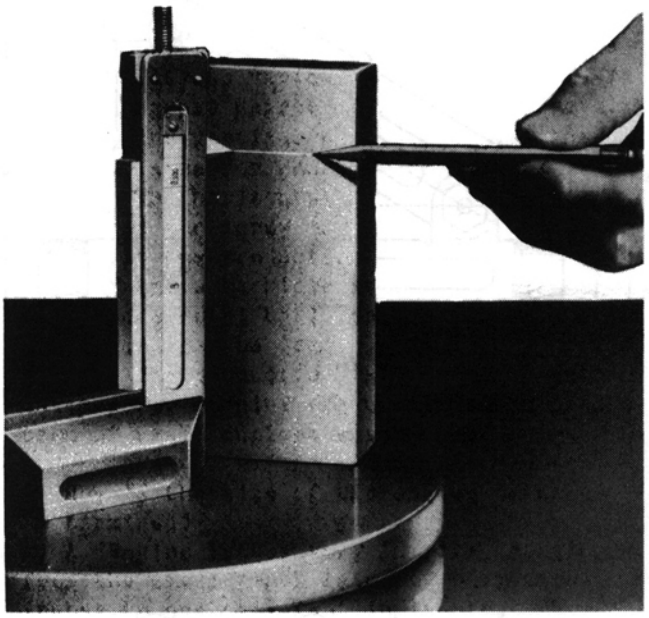


Fig. 701. Establishing Locations Using Accessories

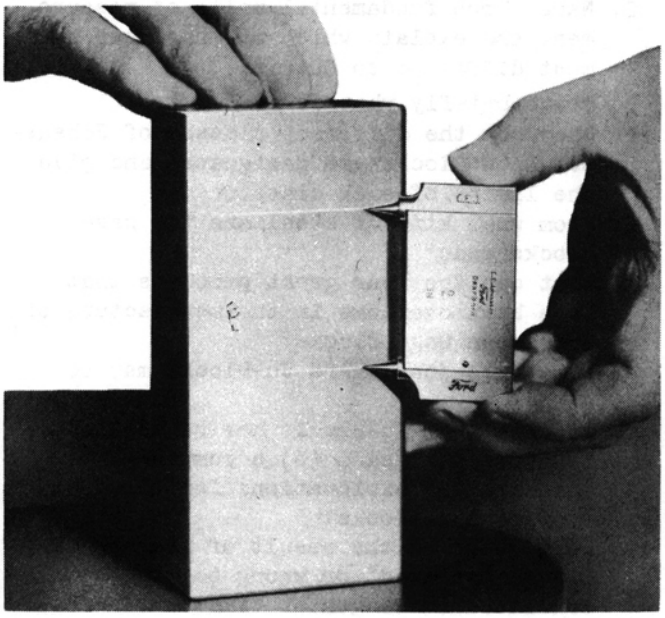


Fig. 702. Rechecking Locations Already Established

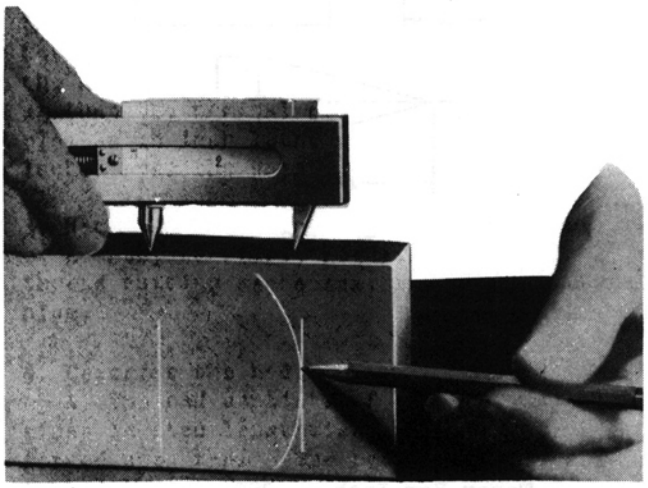


Fig. 703. Establishing a Radius Within Close Limits

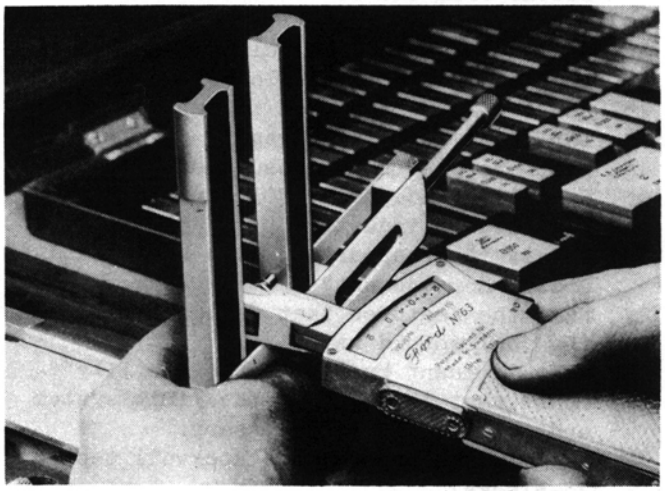


Fig. 704. Setting Internal Indicator Gage for Size

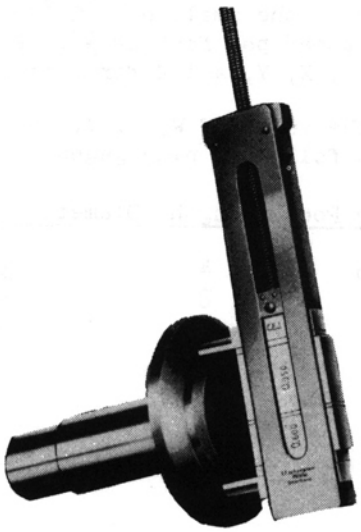


Fig. 705. Checking Outside Diameter of a Plug Gage

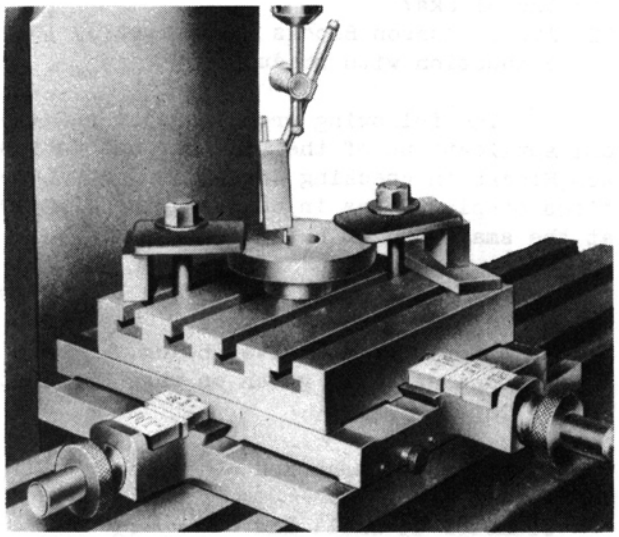


Fig. 706. Setting a Compound Slide by Use of Blocks

1. What is an inch?
2. Name three fundamental units of measurement and explain which one has been the most difficult to obtain.
3. State briefly what Jo-Blocks are.
4. Show how the different classes of Johansson Gage Blocks are designated and give the limits of each class.
5. From what kind of steel are the gage blocks made?
6. What are the four great problems that have been overcome in the manufacture of Johansson Gage Blocks?
7. Name five jobs where Jo-Blocks may be used.
8. Describe how Jo-Blocks may be used as (a) a height gage, (b) a compass.
9. What are the explanations for the cohesion of the blocks?
10. What would be the result of leaving the blocks "stacked" or wrung together for a considerable length of time?
11. Give at least three combinations of blocks from one set that will make .9243".
12. What rules should be followed in building combinations of gage blocks?
13. How are Jo-Blocks used to set a comparator or an amplifying gage?
14. What are precision measurements?
15. How is the accuracy of Johansson Blocks regarded in industry?
16. Why is it necessary to have some standard of measurement?
17. Name the features that an ideal measuring tool should have.
18. What is meant by "interchangeability"?
19. State what is meant by the limit system and tell why it is necessary.
20. What is the approximate Rockwell test of Jo-Blocks?
21. Does the temperature affect the size of the blocks?
22. Are Johansson Blocks used directly in connection with production?

The following problems show practical applications of the sine bar and Johansson Blocks in checking angles, and applications of plug gages in checking the diameter at the small end of a taper.

1. Figure dimensions W, X, Y, and Z used to check a #6 Jarno plug gage. The Jarno taper formulas are given below. These formulas are also given on page 85, Chap. 13, Tapers.

$$\text{Diameter at Large End} = \frac{\text{No. of Taper}}{8}$$

$$\text{Diameter at Small End} = \frac{\text{No. of Taper}}{10}$$

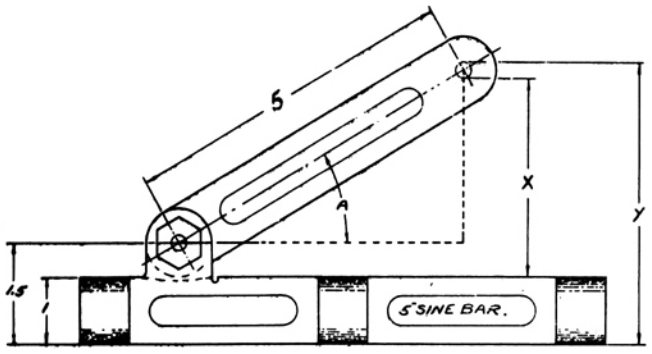


Fig. 707

A = Included angle at which the sine bar is set
X or Y = Combination of Jo-Blocks used to set sine bar at the included angle

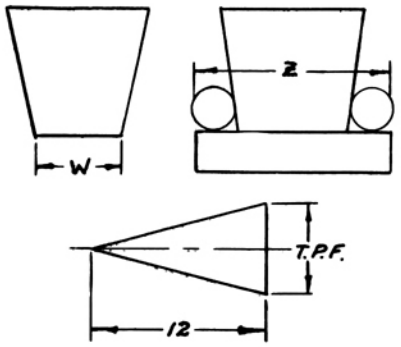


Fig. 708

W = Diameter at small end of taper
Z = Diameter over plugs at small end of taper

$$\text{Length of Taper} = \frac{\text{No. of Taper}}{2}$$

$$\text{Taper Per Foot} = .600$$

$$\text{Taper Per Inch} = .050$$

2. A #7 B & S taper plug gage is .600 in diameter at the small end, 2-7/8 long, and has a taper per foot of $\frac{1}{2}$". Find the value of W, X, Y, and Z for this gage.

3. Find the value of W, X, Y, and Z in each of the following plug gages.

Taper Per Foot	Length	Diameter at Large End
7/16	4	3.250
3/4	8	3.000
7/8	3	2.125
1/4	2	.289

Chapter 23

HEAT TREATMENT

1. What is steel?

A. Steel is an alloy of iron and carbon. Among the other elements found in steel are silicon, phosphorus, sulphur, manganese, and chromium.

2. How may steels be classified?

A. Steels may be roughly classified as straight carbon steels and alloy steels. A straight carbon steel is a steel that owes its properties chiefly to various percentages of carbon without substantial amounts of other alloying elements. When steel contains about .20% or less it may be called a low carbon steel, from .20% to about .60% a medium carbon steel, and from about .60% to 1.30% a high carbon steel. An alloy steel is a steel to which some element other than carbon has been added to improve or change the physical properties.

3. How are the different types of steel designated?

A. In Ford Motor Co. some types of steel are designated by letter. For example, "A" steel is a low carbon alloy steel; "AAA" is a medium carbon alloy steel; "RR" is a high carbon tool steel; etc. Other types are designated by name such as: chrome nonshrink, Ford hot work, insert die steel, etc. Bar stock is identified by the colors painted on the end of the bar (see steel analysis chart, page 204). Many other firms use the S A E numbering system for identifying steel.

4. What is a low carbon steel?

A. A low carbon steel is a steel that does not contain enough carbon to cause it to harden to any great extent when heated to a certain temperature and quenched in oil, water, or brine. It may be heat treated to increase its strength or case hardened to increase its resistance to wear. Some of the low carbon steels are: machine steel, cold rolled steel, and "GG" steel. Among the jobs made from low carbon steel are: clamps, pressure pads, stripper plates, bolts, nuts, washers, and similar jobs where surfaces are not subjected to continuous wear. When hard wearing surfaces and a soft core are required, low carbon steel may be case hardened either by cyaniding or by carburizing and hardening.

5. What is a medium carbon steel?

A. A medium carbon steel is a steel that contains from about .20% to about .60% carbon. "E", "EE", "EEE", "FFF", "H", and "L" steels are medium carbon steels (see steel analysis chart, page 204). Medium carbon steels are used for a wide variety of work including nuts, bolts, stock guides, clamps, flask pins, crankshafts, crane shafts, etc. Medium carbon steels are also used extensively in production work. When heat treatment is required, these steels may be cyanided, pack hardened, or quenched directly from the furnace.

6. What are high carbon steels?

A. High carbon steels are steels which usually contain from about .70% to 1.30% carbon. Tool steel is a high carbon steel. "RR" steel, which contains .95% to 1.05% carbon, is the tool steel most commonly used in Ford Motor Co. Many of the tools and working parts of machines, guide pins, rest buttons, locating pins, dies and punches, gages, bushings, centers, etc., are made from tool steel. Tool steels contain sufficient carbon to cause them to harden; therefore, these steels can be hardened and tempered. The heat treatments used are "Q", "QQ", and "SS". (See standard heat treatment chart, page 207).

7. What are alloy steels?

A. Alloy steels are those steels which contain some alloying elements, such as: chromium, vanadium, nickel, molybdenum, tungsten, etc., in addition to carbon and iron which give them some peculiar characteristic not possessed by ordinary steel. Alloys are put into steels for the following reasons: to secure greater hardness, to secure greater toughness or strength, to enable the steel to hold its size and shape during hardening, or to enable the steel to retain its hardness at high temperatures.

Chromium is used as a hardener, nickel for strength and toughness, tungsten and molybdenum as hardeners and heat registers, manganese for strength, and vanadium as a

purifier and fatigue resister. "A" (low carbon alloy), "AAA" (medium carbon alloy), chrome non-shrink, high speed, Ford hot work, rustless 18-8, insert die steel, and vanadium tool steel are some of the alloy steels used by Ford Motor Co.

8. What jobs are made from "A" and "AAA" steel?

A. "A" steel is used for heavy duty bushings, large worms, large worms where hard surfaces are more important than accuracy, and for thin sections and intricate parts. To obtain a hard surface, "A" steel must be carburized, hardened, and tempered. "AAA" steel is used for acme screws, milling machine arbors, grinder spindles, lock nuts, tap holders, small gear and worms, machine parts, boring bars, and similar jobs (see heat treatment chart, page 207).

9. What is high speed steel?

A. High speed steel is an alloy steel used to make tool bits, forged cutting tools, milling machine cutters, forming tools, reamers, broaches, and similar jobs. It contains a high percentage of tungsten, chromium, vanadium, and carbon. (See analysis chart, page 204). These elements give the steel the ability to retain sufficient hardness at high temperatures to cut metal.

10. Briefly explain the purpose of some of the operations used in heat treatment.

A. Normalizing is a uniform heating above the usual hardening temperatures followed by cooling freely in air. This treatment is used to put the steel back in a normal condition after forging or after improper heat treatment.

Annealing is a uniform heating above the usual hardening temperatures followed by cooling as desired. Annealing may be done either to soften a piece that is too hard to machine or to remachine a piece that has already been hardened. Annealing is also done to relieve internal strains set up by extensive machining. In annealing, the work should be cooled slowly either in the furnace, in a box, or in air.

Quenching is the operation of cooling rapidly in a suitable medium such as water, brine, oil, or air.

Tempering (Drawing) is the operation of reheating a piece that has been hardened to relieve the internal strains and to increase the toughness. All tempering is done below the critical range. The best way to insure correct tempering of a part is to immerse it in a bath of oil or nitrate, in a pyrometer controlled pot like that shown in Fig. 709. Another method is to watch the

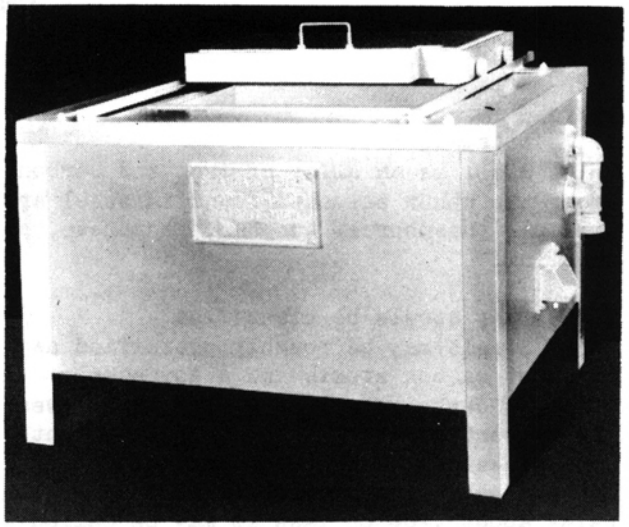


Fig. 709

color of the steel as it heats up in an open furnace. The chart gives the approximate

Color and Temperature Chart

NO.	COLOR	TEMPERATURE
1	Straw	360 to 375° F
2	Light Straw	420
3	Medium Straw	460
4	Dark Straw	490
5	Purple	525
6	Blue	550 to 560
7	Pale Blue	600

temperatures for the various colors.

Strain draw is a low oil draw used on hardened steel immediately after it is quenched to relieve strain and reduce brittleness caused by the shock of sudden quenching. The usual temperature used to strain draw tool steel is 375° to 400° F.

Cyaniding and Carburizing. Low carbon steels (not over .20% carbon) do not become hard when heated above their critical points and quenched, so surface hardening is resorted to for machine and cold rolled steels. Cyaniding is done by keeping the piece immersed in a molten bath of sodium cyanide from 5 to 30 minutes, depending on

size of piece and depth of penetration desired. The piece is then quenched in water, brine, or oil, and a very hard "skin" or casing, .010" to .015" thick is formed. (See Heat Treatment "O", page 208.) This is called "case hardening." Another method used is carburization. The work is placed in a metal box containing a mixture of bone, leather, charcoal, and carburizing materials. The lid is sealed with fire clay and the box is placed in a furnace for some hours at a temperature of 1700° F. The depth to which the carbon penetrates depends upon the length of time the piece is left in the furnace. When properly heat treated, the piece has a hard outer shell and a soft core. (See Heat Treatment "A" and "P", pages 207 and 208.)

11. What is meant by critical points?

A. The critical points or critical temperatures are the temperatures at which some definite change takes place in the physical properties of the steel. These points are important because, in heat treating a piece of steel, it must be heated to a temperature above the upper critical point and then quenched. When the critical points for a certain steel are known, the heat of the furnace can be regulated by a pyrometer. Fig. 710 shows a type of pyrometer used on an electric furnace.

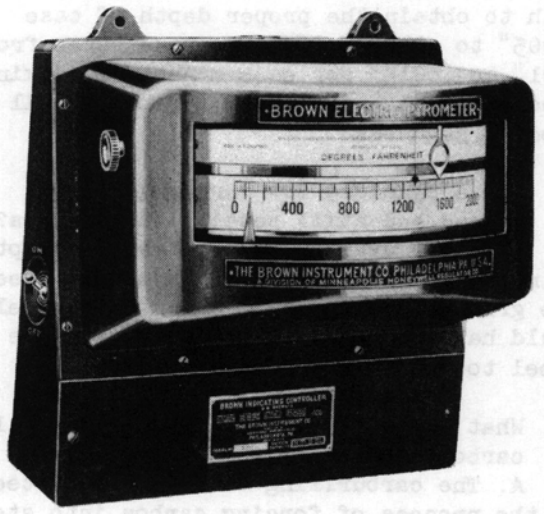


Fig. 710

Not so many years ago it was the custom for a hardener to watch the color of the work in the furnace to determine its temperature. A cherry red, perhaps, for tool steel and an orange, or even at times a lemon yellow for high speed, was thought to be the correct heat. There was a big element of chance in this procedure and the U. S. Bureau of Standards has demonstrated conclusively that at temperatures around 2000° F. the "old timers" who depended on their eyes were off as much as 200° F. in judging furnace temperature. Therefore, the thermocouple and pyrometer have come into use for the accurate measuring of temperatures.

12. What is meant by heat treatment of a metal?

A. Heat treatment is a method by which the heat treater is able to change the physical properties of a metal. There are three major steps in the heat treating of steel: hardening, tempering, and annealing. The hardening operation consists of heating the steel above its critical range and then quenching it in a suitable medium, such as water, brine, or oil. Then it must be given a tempering or drawing treatment which consists of reheating the hardened steel to a temperature below the critical range that will produce the physical properties desired.

The important factors in hardening are: heating to the correct temperature, holding at heat for correct length of time, selecting correct quenching medium and proper method of quenching, and selecting correct drawing temperature. To harden a piece of steel successfully it must be in the softened or annealed state.

13. Name three types of furnaces used in heat treating metals.

A. Gas, oil, and electric furnaces are the most commonly used furnaces. The heat can be easily controlled in these furnaces, which is an important factor. Some steels are heated in open furnaces while others are heated in baths of lead, cyanide of potassium, etc. Fig. 711 shows a pot furnace. If the pot contains molten lead it is called a "lead pot" but if it contains molten cyanide it is called a "cyanide pot." The pot furnace can also be used for tinning baths, for melting low fusion metals, and for other purposes. Tools such as dies, punches, springs, and other small steel parts may be hardened uniformly in this furnace without danger of oxidizing the steel. The lead pot is especially adapted for jobs where only a portion of the tool or part is to be hardened. Only the portion to be hardened is immersed in the lead. The pot furnace is rapid, convenient, and satisfactory.

The type of furnace in which the steel is heated by gas is shown in Fig. 712.

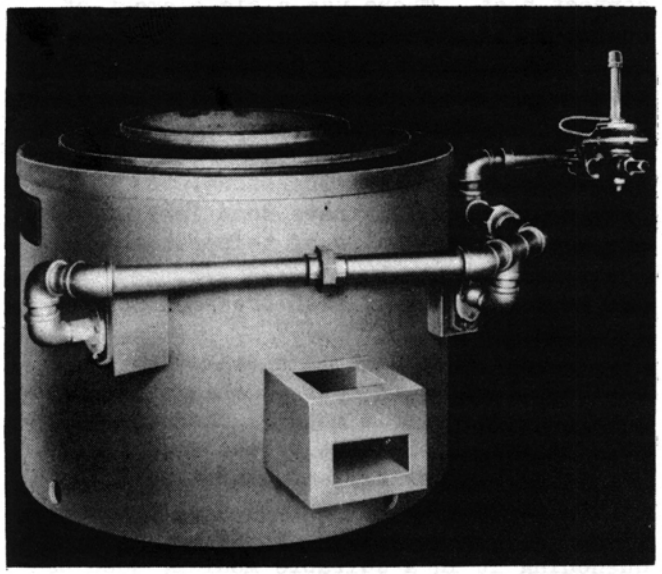


Fig. 711

This is well adapted for most heat treat work. Fig. 713 shows the type of furnaces used for carburizing or annealing.

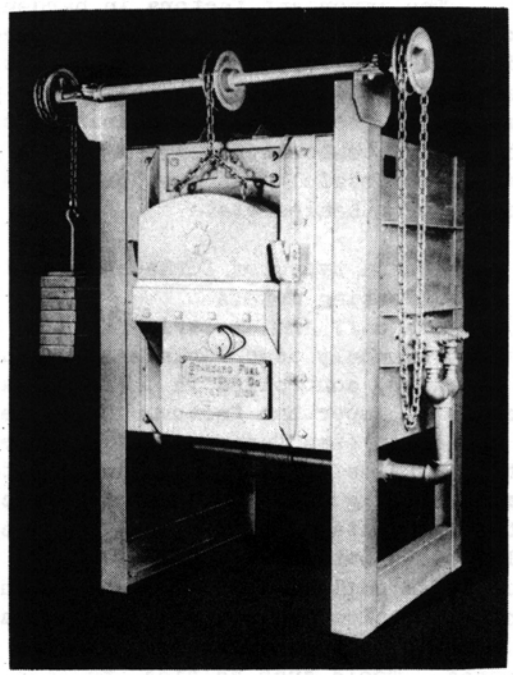


Fig. 712

14. What operations are involved in hardening?

A. Hardening involves both heating and cooling operations.

(a) Heating is the bringing of the steel to the desired temperature above the critical range in order to get the grain structure in the steel into the proper state for hardening.

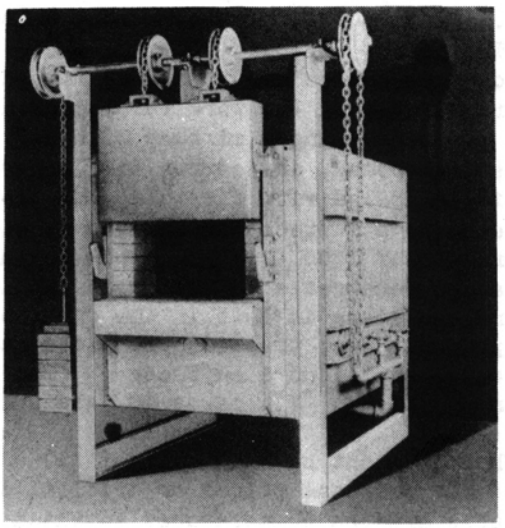


Fig. 713

(b) Cooling is the quenching of the steel in some medium such as water, brine, caustic solution, or oil in order to preserve the structure obtained in heating. The medium must have an even temperature.

15. How are low carbon steels cyanided?

A. Low carbon steels are heated to the required temperature in a sodium cyanide bath to obtain the proper depth of case (.005" to .012"). Cyanide penetrates from .001" to .0015" for each minute of soaking. Pieces are then quenched in water or oil to produce hardness.

16. Can a cyanided low carbon steel be ground and still retain its hardness?

A. Since cyanide penetrates to a depth of not more than .015" on low carbon steel, the grind stock on a job using this steel would have to be less than .015" for the steel to retain any of its hard case.

17. What is meant by the carburizing of low carbon steels?

A. The carburizing of low carbon steels is the process of forcing carbon into steel by packing it in charcoal, bone dust, or other carburizing material and heating it to a temperature of 1700° F. in a furnace like the one shown in Fig. 713. The length of time the furnace is held at this temperature is governed by the depth of the case required. After the steel is removed from the furnace and cooled to room temperature, it can be normalized by reheating from 1560° to

1650° F. and cooling in air. It then can be hardened by inserting into a furnace or a lead pot (see Fig. 711), heating to the required temperature, and quenching.

18. For what is a carburized piece of steel recommended?

A. This steel is recommended for work requiring a hard surface and a tough core. For example, the wrist pin of the Ford car is carburized. This gives a hard surface to resist wear and a tough core to absorb the shock. Many jobs in the toolroom require this treatment. This steel can also be used for spot hardening, as in the nut shown in Fig. 714. In this nut the O D must be hard and the threads must be soft. The operations are as follows. Finish the outside

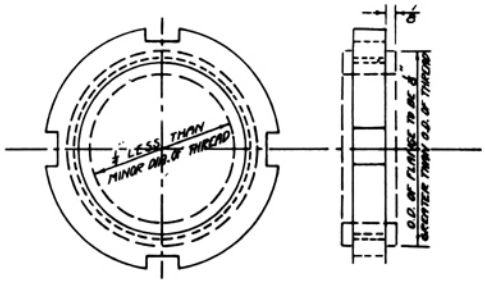


Fig. 714

diameter and the thickness to size, leaving a flange on each side 1/8" greater than the major diameter of the thread and extending 1/8" on each side (see sketch). Bore the hole for the thread 1/4" less than the minor diameter. Mill the slots, carburize, bore the hole to within 1/16" of size, face off the flanges (this removes carbon), harden, and then finish the threads.

19. In the sketch of Fig. 715 a steel block 1" x 4" x 8½", hardened and ground with two holes reamed in place, is required. What kind of steel should be used; what heat treatment is required; and how should this job be done?

A. Use machine steel with heat treatment "P" (a carburizing heat treatment). Use stock 1/4" wider than the width required and machine 1/8" off the two sides marked "A", excepting the square buttons which should be 1/4" to 5/16" larger than the diameter of the holes. The job is then ready to be carburized. After carburizing, machine off the buttons, harden, grind, drill, and ream the holes to size. Machining off the buttons after carburizing and before hardening leaves those spots soft and immune from hardening. The holes may be drilled and reamed after the rest of the piece is hardened, thereby insuring their correct position after hardening. Bear in mind that the holes will be soft.

20. What is meant by nitriding?

A. Nitriding is a method of putting an extremely hard surface on a steel part. The process consists of exposing the steel to hot ammonia gas for some hours. The ammonia breaks down into nitrogen and hydrogen, because of the heat, and the nitrogen reacts with the steel to form the nitride case.

21. Of what importance is the element carbon in straight carbon or "simple" steel?

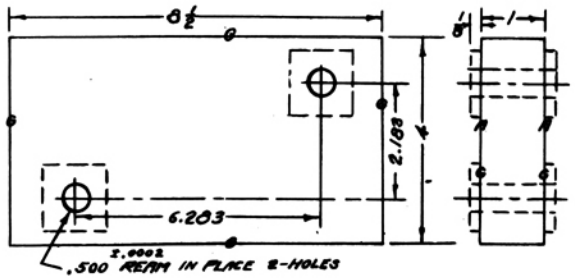


Fig. 715

A. Carbon is the element which causes straight carbon steel to harden.

22. Explain why cold rolled steel, machine steel, or type "A" steel cannot be hardened in the furnace or lead pot to a high degree of hardness.

A. These steels do not contain enough carbon to give a high degree of hardness, although they can be hardened to some extent.

23. How much carbon must be present in steel before it can be hardened noticeably?

A. Steel must contain at least .20% carbon before it can be hardened sufficiently for commercial use.

24. Give the percentage of carbon in each of the following steels: cold rolled, machine, "A", "AAA", "EE", "S", and RR (tool steel).

A. The following list gives the percentage of carbon in each of these steels.

Cold rolled	.05% to .15% carbon
Machine	.08% to .20% "
"A"	.20% to .24% "

"AAA"	.30% to .35% carbon
"EE"	.35% to .40% "
"S"	.60% to .70% "
"RR" (Tool)	.95% to 1.05% "

25. Why is charcoal kept on top of the lead in a lead pot?

A. Charcoal is kept on top of the molten lead in a lead pot to burn up the oxygen in the air, to prevent oxidation, and to keep the job clean. This prevents surface or skin softness and helps to eliminate scale.

26. What heat treatments are used for "AAA" steel? (See heat treatment chart, page 207.)

A. Heat treatments "B" and "C" are used on "AAA" steel. Heat treatment "B" is a cyanide treatment and "C" is a toughening treatment. When heat treatment "C" is called for, be sure to allow for stock from 1/16" to 1/8" to be removed after heat treatment. Heat treatment "C" is given to a piece where accuracy is the most important. It is machinable after this heat treatment.

27. What effect does heat treatment "B" and heat treatment "C" have on the hardness of "AAA" steel?

A. The Rockwell reading on annealed "AAA" steel is from 10 to 15. After heat treatment "C" is given, the reading will increase to from 30 to 35, and after heat treatment "B" is given, the reading should be from 48 to 52.

28. What kind of steel is usually used to make tool bits, milling cutters, reamers, drills, broaches, etc.?

A. High speed steel is usually used for making these tools.

29. Which is the most expensive: machine steel, tool steel, or high speed steel?

A. High speed steel is the most expensive because it contains expensive alloys. Tool steel rates next and machine steel is cheapest. High speed steel costs at least six or seven times as much as tool steel.

30. Why are high speed and chrome non-shrink steels preheated?

A. Some steels such as high speed and chrome non-shrink have a close, fine grain structure. They cannot stand much change in shape when cold, and will crack if put directly into a hot furnace. These steels are heated to a point slightly below the critical range and held there until thoroughly heated before being exposed to the hardening temperatures.

31. How is the hardness of steel determined?

A. The file test and the Brinell,

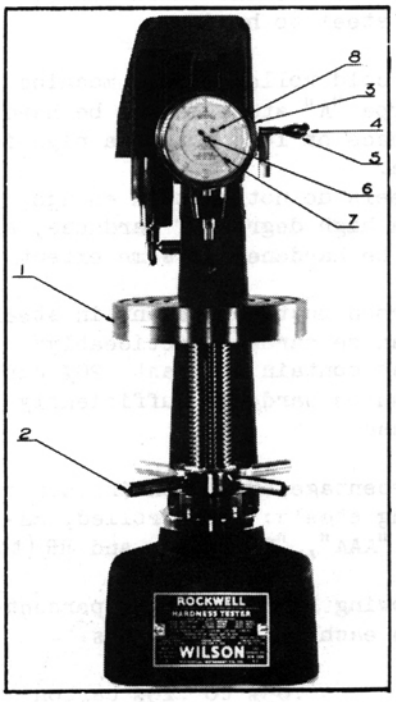


Fig. 716

Instructions for Operating
Rockwell Tester

1. Place piece to be tested upon anvil or testing table.
2. Turn wheel to elevate work into contact with test point and continue turning and forcing work against penetrator till an index shows that Minor Load is applied.
3. Turn bezel of gauge to set dial zero behind pointer.
4. Push handle back an inch to release and apply Major Load.
5. Pull handle forward, removing thereby the Major but not the Minor Load.
6. Observe when moving pointer comes to rest, then--
7. Read ROCKWELL HARDNESS Number on the dial.

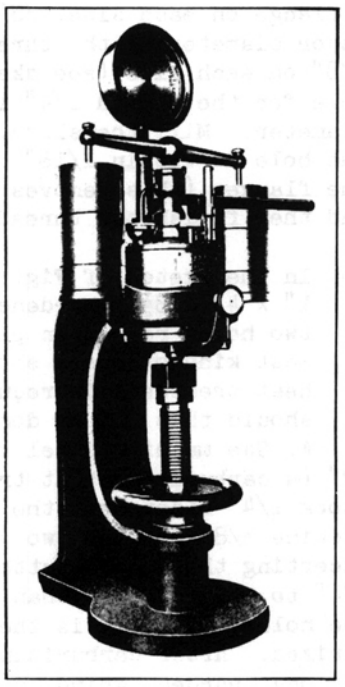


Fig. 717

Rockwell, and Scleroscope hardness testing machines are used to determine the hardness of steel.

The file test is simply trying to cut into the piece with the corner of a file. The hardness is shown by the bite the file will take. This is the oldest and still one of the most useful methods of checking hardness. While this test will not give very definite results because a new file will cut better than an old file and a fine file will make a piece appear softer than a coarse file, it will give results from quite soft to glass hardness. The greatest objection to the use of the file test is that no accurate record of results can be maintained.

In the Rockwell hardness test a 120° diamond cone for hard metals, or a 1/16" steel ball for the softer materials, is impressed into the surface to be tested by a dead weight acting through a series of levers and the depth of penetration measured. The softer the piece the deeper will be the impression under a given load. The average depth of penetration on the softest steel is only about .008 of an inch. The hardness is indicated on a dial gage graduated in the Rockwell "B" and the Rockwell "C" hardness scales. The harder the piece the higher the Rockwell number will be. For example, steel should not show a reading of more than from 30 to 35 on the Rockwell "C" scale to be machinable while a hardened high speed cutter would show a reading of from 63 to 65. When testing hard steels, the diamond point should be used and should be read on the "C" scale; for non-ferrous metals the steel ball should be used and read on the "B" scale. Fig. 716 shows the Rockwell tester and the directions for using it. The average time for each test on a production basis is about five seconds.

In the Brinell test the hardness of the material tested is determined by the resistance it offers to the penetration of a steel ball under pressure. The Brinell hardness number is found by measuring the distance the ball is forced into the piece tested under a given pressure. The greater this distance, the softer the work, and the higher the Brinell number. The width of the indentation is measured with a microscope and the hardness number (corresponding with this width) is found by consulting a standard chart. The Brinell tester (Fig. 717) is most useful in testing soft and medium hard materials and for testing large pieces. On hard steel the impression is so small that it is difficult to read.

In the Scleroscope test a diamond pointed hammer is dropped through a guiding glass tube onto the test piece and the rebound checked on a scale. The harder the steel the higher the hammer will rebound, because the rebound is directly proportional to the resilience or springiness of the test piece. For example, if a ball is thrown on the grass it will not rebound as far as though it were thrown against the sidewalk with the same force. The Scleroscope (Fig. 718) is portable and can be used to check pieces that are too big to place on the anvil of other machines.

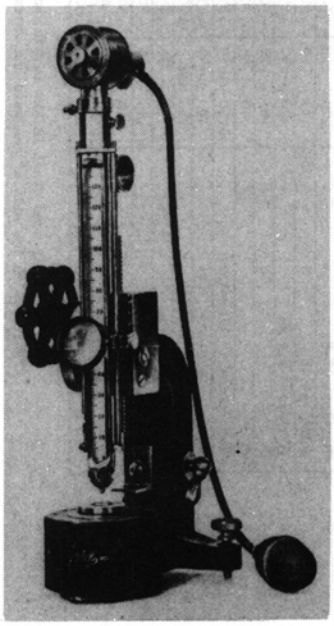


Fig. 718

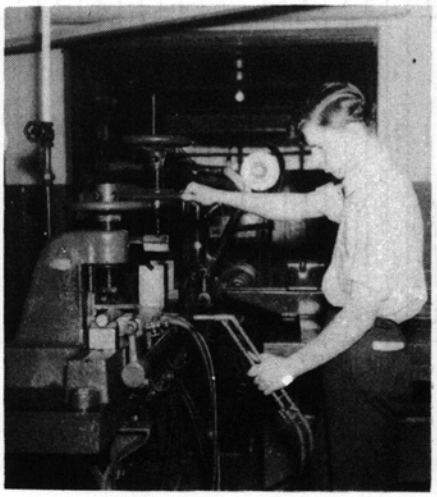


Fig. 719

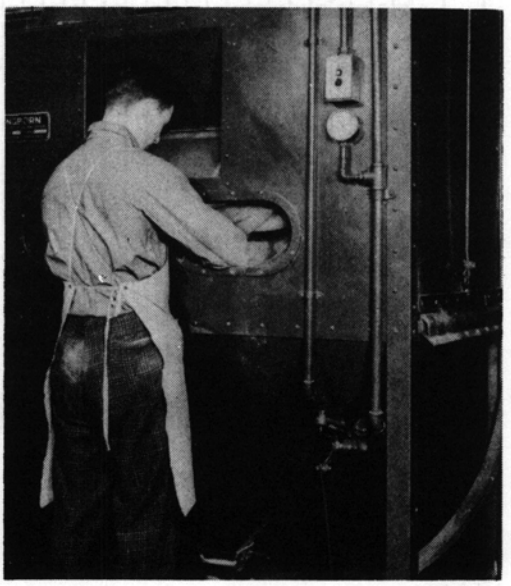


Fig. 720

STEEL ANALYSIS CHART

TYPE OF STEEL	CARBON	MANGANESE	CHROMIUM		SILICON	PHOSPHORUS	SULPHUR	ENDS PAINTED
*A	.20 - .24	.60 - .75	.65 - .80	VANADIUM .12 - .15	.10 - .15	.03 Max	.04 Max	"
AX	.18 - .22	.65 - .75	.80 - .95		.10 - .20	.03 Max	.04 Max	White and Black
AA	.26 - .30	.65 - .80	.80 - 1.00		.10 - .20	.03 Max	.04 Max	Red and Black
*AAA	.30 - .35	.65 - .80	.90 - 1.10		.10 - .20	.03 Max	.04 Max	Red and White
AAAH	.35 - .38	.65 - .80	.90 - 1.10		.10 - .20	.03 Max	.04 Max	Red, White and Blue
AAAAL	.38 - .42	.65 - .80	.90 - 1.10		.10 - .20	.03 Max	.04 Max	Red, Green and Yellow
AAAA	.42 - .47	.70 - .90	.85 - 1.10		.10 - .20	.03 Max	.04 Max	Red and Green
*AAAAA	.48 - .52	.70 - .90	.85 - 1.10		.10 - .20	.03 Max	.04 Max	Blue and White
AA Select	.28 - .32	.65 - .80	.80 - 1.00		.10 - .20	.03 Max	.04 Max	T-498 All Elements
AAA Select	.32 - .35	.65 - .80	.90 - 1.10		.10 - .20	.03 Max	.04 Max	T-12 Well Within Limits
Armature Steel	.05 Max	.30 Max			.12 - .28	.03 Max	.04 Max	
B	.95 -1.05	.20 - .30	.40 - .50		.20 - .30	.03 Max	.04 Max	
BB	.95 -1.05	.20 - .30	.90 - 1.10		.20 - .30	.03 Max	.04 Max	
BBB	.95 -1.05	.30 - .40	1.25 - 1.50		.20 - .30	.03 Max	.04 Max	Blue and Red
Bessemer #1	.09 - .13	.70 - .90				.09-.13	.08-.15	Aluminum
C Key Stock (S.A.E. 1035)	.30 - .40	.50 - .80			.10 - .20	.05 Max	.05 Max	
C Pure Iron	.03 Max	.12 Max		COPPER OPTIONAL		.01 Max	.04 Max	
*Chrome Non-Shrink	1.45 -1.60	.25 - .35	11.00 - 12.00	V .20-.25 Mo .70-.90	.20 - .40	.03 Max	.04 Max	
*CC	.44 - .50	.40 - .55	.70 - .90	TUNGSTEN 1.00 - 1.20	.15 - .25			
D	.45 - .52	.80 - .95	1.00 - 1.20		.10 - .20	.03 Max	.04 Max	Green With White Stripe
DD	.48 - .52	.80 - .95	1.00 - 1.20		.10 - .20	.03 Max	.04 Max	Green
*Die Block Ajax	.60 - .75	.30 - .40	3.25 - 3.75		.10 - .20	.03 Max	.04 Max	
*Die Block Hammer	.47 - .55	.50 - .60	.60 - .75	NICKEL 1.50 - 1.75	.10 - .20	.03 Max	.04 Max	
E	.27 - .35	.70 - .90			.07 - .15	.04 Max	.05 Max	Red
EE	.35 - .40	.70 - .90			.07 - .15	.03 Max	.05 Max	Yellow
*EEE	.40 - .45	.70 - .90			.07 - .15	.03 Max	.05 Max	Red and Yellow
Electrical	.05 Max	.30 Max			.90 -1.20	.03 Max	.03 Max	
F	.08 - .15	.80 - .99			.10 - .20	.04 Max	.08-.15	Aluminum and Black
FF	.15 - .20	.80 - .99			.10 - .20	.04 Max	.10-.15	Aluminum and Green
FFF	.34 - .40	.80 - .99			.10 - .20	.04 Max	.10-.15	Aluminum and Yellow
Ford High Speed (Taps and Drills)	.65 - .73	.25 - .35	3.75 - 4.25	V 1.00-1.25 W 17.0-18.0	.20 - .30	.03 Max	.04 Max	
*Ford Hot Work	.18 - .23	.40 - .60	1.40 - 1.60	MOLYBDENUM .45 - .55	.15 - .25	.04 Max	.04 Max	Red and Green With White Stripe
*Ford Special High Speed	.78 - .84	.25 - .35	4.00 - 4.50	V 2.00-2.25 W 18.0-19.0	.20 - .50	.03 Max	.04 Max	(Mo. = .50 - .80)
G	.08 - .15	.30 - .45			.07 - .15	.03 Max	.05 Max	Black
*GG	.15 - .20	.30 - .45			.07 - .15	.04 Max	.05 Max	Black
H	.27 - .37	.45 - .60			.07 - .15	.04 Max	.05 Max	Blue
Key Stock (S.A.E. 2330)	.25 - .35	.50 - .80		NICKEL 3.25 - 3.75		.04 Max	.045 Max	
L	.23 - .30	.35 - .50				.04 Max	.05 Max	Green and Blue
Low Carbon, Open Hearth	.05 - .15	.30 - .50				.04 Max	.05 Max	Black With Yellow Stripe
*Machine	.08 - .20	.35 - .50			.10 - .20	.04 Max	.05 Max	Black
Magnet	.82 - .90	.30 - .45	2.25 - 2.60	(High Limit Preferred)	.25 - .40	.03 Max	.04 Max	Aluminum When C is 185 or over
N	.12 - .16	.35 - .45	.30 - .40		.10 - .20	.03 Max	.04 Max	Green and Black
R	.70 - .80	.20 - .35	.10 Max	NICKEL - NONE	.15 - .25	.025 Max	.03 Max	
*RR	.95 -1.05	.20 - .35	.10 Max	NICKEL - NONE	.15 - .25	.025 Max	.03 Max	Brown
RRR	1.20 -1.30	.20 - .35			.15 - .25	.025 Max	.03 Max	
*Rustless 18-8	.05 - .10	.30 - .45	16.0 - 18.0	NICKEL 7.0 - 9.0	.15 - .30	.04 Max	.05 Max	
Rustless 18	.05 - .10	.30 - .45	16.0 - 18.0		.50 Max	.04 Max	.05 Max	
Rustless Type II	.20 - .30	.25 - .40	12.0 - 14.0		.70 -1.00	.04 Max	.05 Max	
*S	.60 - .70	.70 - .85			.15 - .20	.03 Max	.04 Max	
*SS	.70 - .85	.70 - .85			.10 - .20	.03 Max.	.04 Max	Blue and Yellow
Tap (Under 1" Diameter)	1.20 -1.30	.25 - .40	.35 - .45	V .15-.25 W 1.25-1.50	.30 - .45	.025 Max	.025 Max	
V	.35 - .45	.25 - .40	1.85 -2.50		3.60 -4.20	.03 Max	.04 Max	
Welding Wire	.10 Max	.20 Max				.02 Max	.03 Max	
*Insert Die Steel	.55 - .60	.45 - .60	.70 - .80	Mo .75-.80 Ni 2.25-2.45		.03 Max	.04 Max	
*Vanadium Tool Steel	.95 -1.05	.20 - .35	.10 Max	V .40-.50 Ni None	.15 - .25	.025 Max	.03 Max	
S.A.E. #4620	.18 - .22	.30 - .60	.25 Max	Mo .20-.30 Ni 1.65-2.00	.15 - .30	.04 Max	.05 Max	Aluminum and Red
Deep Drawing Steel	.05 - .08	.28 - .38	.04 Max	Copper .10 Max		.03 Max	.04 Max	
"FFH"	.20 - .25	.80 - .99			.10 - .20	.04 Max	.10-.15	

*Most commonly used steel in the school shop.

"White if on low side of analysis and white with green stripe if on high side.

4-5-38

32. Give some suggestions that are valuable in heat treatment.

A. Work having sharp edges or different cross sections should be protected by pieces of wire and fire clay to prevent fractures.

Cold rolled steel is a low carbon steel that can be case hardened by immersing in a cyanide pot and quenching. Drill rod is made from "R" (high carbon) steel and can be hardened by heating in a lead pot or a furnace and quenching.

Steel should be heated long enough to insure a good even heat throughout. The practical rule is to keep or "soak" the piece in the furnace one hour for each square inch of cross sectional area.

To avoid warping a long slender piece when quenching, hold it vertically over the bath and plunge straight down. A warped piece may be straightened under pressure in a straightening press after heating with a blow pipe (Fig. 719).

The sand blast (Fig. 720) is used to remove scale and dirt from parts previously heat treated. This saves a great deal of the grinder's time.

33. What are some of the safety rules which must be observed in this department?

A. The following rules must be observed by students in heat treating steel:

(1) GOGGLES MUST BE WORN when working on a lead, cyanide, or nitrate pot.

(2) Do not put anything damp or wet into these pots or an explosion will occur.

(3) Hot tongs should not be left where any one can be burned by them.

(4) Never pick up anything until you know whether or not it is hot.

(5) Rubber gloves must be worn while sand blasting.

(6) Do not wear gloves while grinding.

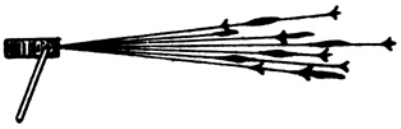
(7) Do not use a towel to hold a piece while grinding.

The "spark test" is a method by which different steels that are not marked may be identified within certain limits by the sparks given off when they are held against a grinding wheel. In this test either a portable or a stationary grinder may be used. Only enough pressure to maintain a steady contact between the work and the wheel is necessary and less pressure is required as the wheel speed is increased. About 8,000 surface feet per minute is usually a satisfactory speed. The sparks from most steels will be practically the same whether the stock is hardened or annealed. Test pieces of known samples on which to practice or with which to compare pieces being checked should be available. The test should be made in diffused daylight if possible.

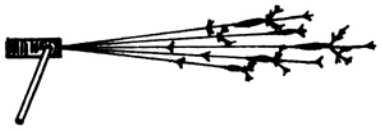
Broadly speaking, the effect of various elements is as follows: Carbon causes the sparks to burst. Manganese tends to brighten the spark and increase the spray around the periphery of the wheel. Chromium darkens the color, suppresses the stream and bursts, and causes fine carrier lines. Nickel suppresses the stream slightly and causes forked tongues. Tungsten suppresses stream and bursts, and causes fine red carrier lines. Molybdenum causes a detached spear head at the end of the ray.

The following chart shows sketches of sparks from some of the most commonly used steels. They are of necessity very general but may be used to point out certain characteristics. For example, the difference in carbon content is indicated by the difference in the number of sparks as shown in the first three sketches. If a piece of machine steel and a piece of high speed steel were sparked, the difference between them could be easily discerned. However, proficiency in identifying all steels can only be gained by practice and experience. The spark test does not analyze a piece of steel, but is simply a guide in identifying it.

SPARK TEST CHART



(a) Low carbon steels show long club-shaped sparks, smooth light lines, light yellow in color, and no small stars. Cold rolled, machine steel, etc., are in this group.



(b) Medium carbon steels show club-shaped lines which split into formation of a few small simple stars of light yellow color. Steels such as "E", "EE", and "EEE", are in this group. The spark from "AAA" steel, which is a medium carbon alloy, is very similar in formation but darker in color due to the chromium content.

(c) High carbon steels such as "RR" (tool steel) show almost no club-shaped sparks, but numerous little light yellow stars. "S" and "SS" steels are included in this group.

(d) Chrome steels show yellow spark lines with ball-shaped ends, between which occur more or less numerous and active sheafs of light depending upon the carbon content. Chrome non-shrink, the rustless steels, etc., are in this group.

(e) High speed steels show several interrupted, dotted, brownish-red spark lines with ball-shaped, end sparks of dark blood-red color. Some of these burst into small red stars of few rays, slightly lighter in color.

Hardness Conversion Table
(Approximate)

Brinell		Rockwell			Brinell		Rockwell		
Diam. 3000 Kg., 10mm. Ball	Hardness No.	C Scale 150 Kg., 120° Cone	B Scale 100 Kg., 1/16" Ball	Shore	Diam. 3000 Kg., 10mm. Ball	Hardness No.	C Scale 150 Kg., 120° Cone	B Scale 100 Kg., 1/16" Ball	Shore
Mm									
2.20	780	68	...	96	3.45	311	32	...	43
2.25	745	67	...	94	3.50	302	31	...	42
2.30	712	65	...	92	3.55	293	30	...	41
2.35	682	63	...	89	3.60	285	29	...	40
2.40	653	62	...	86	3.65	277	28	...	38
2.45	627	60	...	84	3.70	269	27	...	37
2.50	601	58	...	81	3.75	262	26	...	36
2.55	578	56	...	78	3.80	255	25	...	35
2.60	555	55	...	75	3.85	248	24	100	34
2.65	534	53	...	73	3.90	241	23	99	33
2.70	514	51	...	71	3.95	235	22	99	32
2.75	496	50	...	68	4.00	229	21	98	32
2.80	477	48	...	66	4.05	223	20	97	31
2.85	461	47	...	64	4.10	217	18	96	30
2.90	444	46	...	62	4.15	212	17	95	30
2.95	429	44	...	60	4.20	207	16	95	29
3.00	415	43	...	58	4.30	197	14	93	28
3.05	401	42	...	56	4.40	187	12	91	27
3.10	388	41	...	54	4.50	179	10	89	25
3.15	375	39	...	52	4.60	170	8	87	24
3.20	363	38	...	51	4.70	163	6	85	23
3.25	352	37	...	49	4.80	156	4	83	23
3.30	341	36	...	48	4.90	149	2	81	22
3.35	331	35	...	46	5.00	143	0	79	21
3.40	321	34	...	45	5.10	137	−3	77	20

Heat Treatment Chart

COMMERCIAL ANNEALING — HEAT TREATMENT "D"
(1) Heat to 1425° F. (2) Cool in air. Use to restore hardened steel to machining hardness.

COMMERCIAL ANNEALING — HEAT TREATMENT "DD"
(1) Heat to 1500-1550° F. (2) Cool in mica. To eliminate scale, pack in charcoal. Use for U.S. die steel to obtain maximum softness.

CAST IRON — HEAT TREATMENT "CC"
(1) Heat in furnace 1400-1450° F. (2) Cool slowly in furnace or mica. Use for surface plates before machining.

TYPE "A" STEEL — HEAT TREATMENT "A"
(1) Carburize at 1700° F. (2) Cool in carburizing box. (3) Reheat to 1650° F. (4) Cool in air. (5) Reheat to 1560° F. (6) Quench in brine. (7) Strain draw in oil 350-375° F. Rockwell C-62-64. Use when a hard surface and a soft core is the most important requirement. Never use heat treatment "A" when minimum of distortion is desired. Use for heavy duty bushings, large worms, and large gears where a hard surface is more important than accuracy.

TYPE "A" STEEL — HEAT TREATMENT "E"
(1) Carburize at 1700° F. (2) Cool in carburizing box. (3) Reheat to 1650° F. (4) Cool in air. (5) Reheat to 1560° F. (6) Quench in oil. (7) Strain draw in oil 350-375° F. (8) Draw temper if specified. Rockwell 56-64. For thin sections and intricate parts to eliminate distortion.

TYPE "AAA" STEEL — HEAT TREATMENT "B"
(1) Heat in cyanide 1500° F. (2) Soak 5 to 10 minutes. (3) Quench in oil. (4) Strain draw in oil 350-375° F. Specify Rockwell hardness required. Use for acme screw, milling machine arbors, grinder spindles, lock nuts, tap holders, small gears and worms, etc. For large parts, where cyanide is not available, heat in furnace.

TYPE "AAA" STEEL — HEAT TREATMENT "C"
(1) Rough machine. (2) Heat to 1560° F. (3) Quench in brine or caustic solution. (4) Draw to 950-1050° F. (5) Cool in air. (6) Finish machine. Rockwell 30-34. Use for large gears, armature shafts, large boring bars, miscellaneous heavy machine parts, etc., and where accuracy and toughness are more important than hardness.

TYPE "CC" STEEL — HEAT TREATMENT "H"
(1) Heat in lead to 1500° F. (2) Quench in brine or caustic. (3) Draw in oil to 350° F. Rockwell 54-59. Use for chisels, miscellaneous hand tools, and wire machine rolls.

TYPE "E" STEEL — HEAT TREATMENT "AA"
(1) Heat to 1470-1500° F. (2) Quench in brine. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified. Rockwell 43-46.

TYPE "EE" STEEL — HEAT TREATMENT "BB"
(1) Heat to 1470-1500° F. (2) Quench in brine. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified. Rockwell 51-55.

TYPE "EE" AND "EEE" ST C — HEAT TREATMENT "ST"
(1) Preanneal at 1700-1750° F. (2) Cool in air. (3) Heat in cyanide to 1470° F. (4) Quench in solution. Rockwell 33-39.

TYPE "G", "GG", AND LOW CARBON STEEL
Use heat treatments "O" and "P". (See machine steel.)

TYPE "R" (TOOL) STEEL — HEAT TREATMENT "QQ"
(1) Heat to 1450-1475° F. (2) Quench in brine. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified. Rockwell 55-59. For heavy duty shears, dies, and punches.

TYPE "R" (TOOL) STEEL — HEAT TREATMENT "SS"
(1) Heat in lead 1450-1475° F. (2) Quench in brine. (3) Strain draw in oil 375° F. (4) Draw temper if specified. Rockwell as specified. Use for calking tools, drift pins, machine hammers, pin drivers, picks, and screw drivers.

TYPE "RR" (TOOL) STEEL — HEAT TREATMENT "Q"
(1) Heat to 1400-1450° F. (2) Quench in brine. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified. Rockwell as required. Use for chuck jaws, ball races, bushings, cams, female gages, rest buttons, arbors, locators, plug gages, snap gages, "V" blocks, etc. Use when maximum hardness is desired. Temperature to be governed by size of steel. For small parts quench in oil, if the same hardness can be obtained.

TYPE "RR" (TOOL) STEEL — HEAT TREATMENT "S"
(1) Heat in lead to 1400-1450° F. (2) Quench in brine. (3) Strain draw in oil 450° F. (4) Draw temper if specified. Use for partial hardening.

TYPE "S" (SPRING) STEEL — HEAT TREATMENT "SP"
(1) Heat to 1450° F. (2) Quench in oil. (3) Draw temper 750° F. Rockwell 41-44. Use for spring wire $\frac{1}{4}$ diameter and below.

TYPE "SS" (SPRING) STEEL — HEAT TREATMENT "SSP"
(1) Heat to 1450° F. (2) Quench in oil. (3) Draw temper 750° F. Rockwell 41-44. Use for spring wire $\frac{1}{4}$ diameter and above.

Heat Treatment Chart (Continued)

MACHINE AND COLD ROLLED STEEL HEAT TREATMENT "O"

(1) Heat in cyanide 1500-1560° F. (soak 10 minutes). (2) Quench in brine (small parts in oil). File hard. Use for a hard surface not subject to continuous wear. Do not grind. Use for clamps, locating gages, pressure and stripper pads, stock guides, bolts, nuts, and washers.

MACHINE AND COLD ROLLED STEEL HEAT TREATMENT "P"

(1) Carburize at 1700° F. (2) Cool in carburizing box. (3) Reheat to 1650° F. (4) Cool in air. (5) Reheat to 1425° F. (6) Quench in brine. (7) Strain draw in oil 350-375° F. Rockwell 62-64. Due to soft core, should be used for parts difficult to straighten. This heat treatment may also be used for selective hardening where it is necessary to machine after hardening. (1) Leave stock, (2) carburize, (3) remove case, (4) harden, (5) machine. (See questions 17 and 18.)

HIGH SPEED STEEL HEAT TREATMENT "G"

(1) Preheat to 1400-1450° F. (2) Superheat 2350° F. minimum. (3) Quench in oil. (4) Draw in furnace 1050-1100° F. if not otherwise specified. Rockwell 63 minimum. Use for gear cutters, hobs, reamers.

HIGH SPEED STEEL HEAT TREATMENT "J"

(1) Preheat to 1400-1450° F. (2) Superheat 2300-2350° F. (3) Cool in air. (4) Draw in furnace 1050-1100° F. if not otherwise specified. Rockwell 62-64. Use for long keyway broaches, long and slender tools, and to eliminate distortion. To reduce brittleness in heat treatments "G" and "J", redraw in nitrate 700-800° F. for 3 or 4 hours.

HIGH SPEED STEEL HEAT TREATMENT "L"

(1) Preheat 1400-1450° F. (2) Superheat 2350° F. (3) Quench in cyanide 1050-1100° F. (4) Draw in furnace 1050° F. Rockwell 63 minimum. Used only for special forming tools, threads, hobs, and similar tools, where accuracy is important.

HIGH SPEED STEEL HEAT TREATMENT "SPG"

(1) Preheat to 1450° F. (2) Superheat to 2300° F. (3) Cool in air. (4) Preheat to 1450° F. (5) Superheat to 2350-2400° F. (6) Quench in oil. (7) Furnace draw 1050° F. (8) Finish to print. Rockwell C-64 minimum. Use for form tools, broaches, counterbores, spotfacers, milling cutters, cut-off tools, reamers.

HIGH SPEED STEEL (SPECIAL)

Special heat treatment before backing off high speed steel cutters. (1) Heat to 1650° F. (2) Draw 1200° F. (3) Quench in oil. Rockwell 34-36. Sand blast.

SEMI-HIGH SPEED STEEL HEAT TREATMENT "X"

(1) Preheat to 1500° F. (2) Superheat 2050-2100° F. (3) Draw at 1175-1250° F. Rockwell 46-50. Use for hot stock only.

CHROME NON-SHRINK STEEL HEAT TREATMENT "M"

(1) Preheat to 1200-1250° F. (2) Superheat 1860-1880° F. (3) Air cool. (4) Draw 980-1000° F. Rockwell 59 minimum. Use for dies.

CHROME NON-SHRINK STEEL HEAT TREATMENT "N"

(1) Pack in charcoal. (2) Heat slowly to 1850° F. (3) Remove from pack; air cool. (4) Draw 350° F. Rockwell 63 minimum. Use for master gears, gauges, etc. Parts for extreme hardness and wear.

CHROME NON-SHRINK STEEL HEAT TREATMENT "R"

(1) Preheat 1250° F. (2) Superheat 1860-1880° F. (3) Air cool. (4) Draw 750° F. Rockwell 56-58. Use for small details.

CHROME NON-SHRINK ST C HEAT TREATMENT "MM"

(1) Preheat 1250° F. (2) Superheat 1850° F. (3) Air cool. (4) Draw 980-1020° F. (5) Air cool. Rockwell 58-62.

NON-SHRINK STEEL HEAT TREATMENT "V"

(1) Heat to 1400-1450° F. (2) Quench in oil. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified. Rockwell 60-62. Use for intricate parts where minimum of distortion is desired.

NON-SHRINK STEEL HEAT TREATMENT "Z"

(1) Heat in lead 1400-1440° F. (2) Quench in oil. (3) Strain draw in oil 350-375° F. (4) Draw temper if specified.

U.S. DIE STEEL HEAT TREATMENT "F"

(1) Heat to 1560-1600° F. (2) Quench in oil. (3) Remove from oil while hot 600° F. (4) Allow oil to burn off. Use for dies and punches for Ajax and National machines, small miscellaneous parts, and boring bars.

FORD HOT WORK STEEL HEAT TREATMENT "FF"

(1) Heat to 1600° F. (2) Quench in oil. No draw. Rockwell 41-45. Use for dies and punches for Ajax and National Machines, all socket wrenches.

FORD DIE BLOCK NO. 1 HEAT TREATMENT "W"

(1) Heat in furnace to 1400-1450° F. (2) Quench in oil. Rockwell 39-44. Use for hammer dies. Draw 800-850° F., time governed by size of die.

INSERT DIE STEEL HEAT TREATMENT "DS"

(1) Heat to 1620° F. (2) Cool in air (fan blast). (3) Draw to 1050-1075° F. Rockwell 42-46. Use for hammer die inserts, hot heading dies, etc.

Chapter 24

ABRASIVES AND GRINDING WHEELS

HISTORY

AN ABRASIVE is any material having the ability to wear away a substance softer than itself. Sand and sandstone are perhaps the oldest abrasives known to mankind. Prehistoric man used sand and sandstone to form or shape edges of tools by which he earned his living and as tools became more and more important for preservation of life, he became more dependent on natural abrasives for maintenance of sharp tools.

EMERY and CORUNDUM are two "natural abrasives" which were commonly used in industry for the purpose of sharpening edged tools. They occur as a mineral deposit in the earth's crust. These abrasives, formed into wheels, were very superior to the old grindstones in that they were capable of faster cutting and could be made coarse or fine. In spite of this they could not meet the demands of industry because they contained impurities which were difficult to extract and because the percentage of the important cutting element, aluminum oxide, was not constant.

The only other element known to be harder than emery or corundum was the diamond, but its cost was prohibitive as far as industrial usage was concerned. In 1891 Dr. Edward G. Acheson set himself to the task of trying to produce artificial diamonds. His experiment consisted of combining powdered coke and corundum clay at extremely high temperatures. After the mass cooled it was found to contain brightly colored crystals which upon examination proved to possess the ability to cut glass and to have a slight cutting effect on diamonds. To this new substance Dr. Acheson gave the name "Carborundum" because it was formed from carbon and corundum; later however, this new substance was found to be composed of silicon and carbon so that the name "Silicon Carbide," having the chemical symbol SiC, was given to it.

This new material, silicon carbide, was considered as the answer for a better abrasive, but cost and limited methods of manufacture kept it from being used except as a lapping compound for finishing precious jewels; with the development of hydroelectric generators and cheap electric power, the cost of production was cut to a point where all industry could avail themselves of its use.

About the same time that Dr. Acheson was experimentally producing silicon carbide, Charles P. Jacobs, an engineer at Ampere, New Jersey was conducting similar experiments to produce a better grade of emery. His experiments consisted of trying to extract the impurities of sand, iron, and titanium oxides from clay deposits rich in aluminum oxide by use of a small electric furnace. The result was a product which consisted of about 95% pure aluminous oxide, which possessed characteristics very similar to those of silicon carbide.

CHARACTERISTICS AND MANUFACTURE

While these two artificial abrasives are very similar to each other, the properties of them differ widely. Silicon carbide is extremely hard, being rated at 9.87 on the Moh scale of hardness as compared to 10.0 for the diamond. It is easily fractured by impact and its excellence depends upon the purity of the ingredients used in making it. It has a specific gravity of 3.18, which is very low in comparison to other abrasives. Aluminum oxide is not as hard as silicon carbide but it is much tougher. On the Moh scale of hardness it is rated at 9.6 as compared to 9.87 for silicon carbide and 10.0 for the diamond.

SILICON CARBIDE is suited for grinding materials having a low tensile strength but which are very hard in nature, such as ceramics, pottery, tungsten carbide, etc.

ALUMINUM OXIDE, because of its toughness, is resistant to shock and therefore suitable for grinding materials of high tensile strength, such as tool steel, high speed steel, "AAA", etc.

Artificial abrasives have a distinct advantage over natural abrasives in that purity of product and grain size can be readily controlled. This is important because undersize grains cannot do their share of the work while oversize grains give a poor finish to the work.

Electric furnaces are used to produce

GRAIN SIZE CHART

Very Coarse	Coarse	Medium	Fine	Very Fine	Flour Sizes
8	12	30	70	150	280
10	14	36	80	180	320
	16	46	90	220	400
	20	60	100	240	500
	24		120		600

both abrasives. Silicon carbide is made in an open trough-like furnace, by fusing a mixture of coke, sawdust, sand, and salt. After the mass has cooled, the sides of the furnace are let down, exposing a big clinker; this clinker is broken by means of a drop weight, and the pieces put through a crushing machine. As the abrasive particles leave the last crusher, they are magnetically cleaned and washed, after which they pass onto shaker screens. The mesh of these screens vary in size from 4 to 220 meshes to the lineal inch, and by their vibrating action sort the grains according to a definite size. If the abrasive passes through a screen having 30 meshes to the lineal inch, but is retained on a screen of 36 mesh, it is called a #30 abrasive. Abrasives finer than 220 are graded for size by hydraulic or sedimentation methods. After the abrasive has been graded to size, it is dried and placed in storage bins or hoppers for future use. Above is a chart showing how the various grain sizes are classified.

GRINDING WHEELS

Grinding wheels are formed by using a suitable material to cement or bond the abrasive grains together in the desired shape. The hardness or softness of the wheel is dependent on the amount and kind of bonding material used. Since the hardness rating of the abrasive is constant it is apparent that the bond can have no effect whatever upon it. When speaking of the hardness or softness of the wheel, it is always understood to have reference to the strength of the bond.

BONDS

There are a great many different kinds of bonds but the kinds commonly used are the VITRIFIED, SILICATE, SHELLAC, RUBBER, and RESINOID. Of these bonds, the vitrified and silicate are used most. The VITRIFIED bond is used in making about 75% of all grinding wheels. The reason for this is because its strength and porosity enable it to remove considerable stock from the job for each inch of wheel wear; it is not affected by water, acids, or ordinary temperature changes, and it is free from hard and soft spots. In the vitrified process, glass, feldspar, flint, or other ceramic substances are mixed with the abrasive and subjected to heat which causes the bond to form a glass-like structure between each abrasive particle.

In the SILICATE bond, silicate of soda is used. Besides the amount of bond used, the hardness of this wheel is governed by the amount of tamping or pressing. This produces a wheel which is milder acting than the vitrified wheel and permits the abrasive grains to be released more readily, therefore, they do not heat up so fast. Silicate wheels can be made in larger diameters than vitrified wheels and are generally used for grinding edged tools such as drills, reamers, milling cutters, etc.

RUBBER wheels are made of a mixture of abrasive, rubber, and sulphur; the mass is then pressed into shape and given a mild vulcanizing treatment. Wheels of this bond are used for high speed grinding operations and because of their high safety factor can be made very thin and can be used for cutting off steel stock.

SHELLAC bonded wheels are made by mixing the abrasive and bond in a heated machine which mixes and completely coats the abrasive with the bonding material. After the wheels are formed they are placed in an oven, covered with sand, and then baked for a short time at approximately 300° F. Wheels of this bond are used very extensively for grinding mill rolls and jobs where a high luster finish is required.

RESINOID bonded wheels are made by mixing powdered resinoid with the abrasive particles and then adding a plastic substance so that the wheels can be molded. The molding is then placed in an electric oven and heated from a few hours to three or four days depending on the size of the wheel, at approximately 300° F. Upon cooling, the wheel becomes very hard. Wheels of this bond are generally used in foundries for snagging castings or for cleaning up steel billets.

WHEEL STRUCTURE

The amount of puddling or packing that is applied in forming the grinding wheel is very influential in the way the wheel cuts, because it affects the wheel structure (Fig. 721) or grain spacing. Two

Fig. 721

Grain spacing: (a) wide; (b) medium; (c) close. This diagrammatic sketch shows how three wheels may be identical in grain size and grade (bond strength) but differ in grain spacing or STRUCTURE--and thereby differ in grinding action.

wheels of the same grade and grain size but of different grain spacing will have different cutting actions. Wheels with wide grain spacing should be used on hard dense materials and those with close spacing should be used to secure a good finish on the work. For this reason wheel life can often be increased without sacrificing grinding quality, using the same grain and grade of wheel but with a different structure.

WHEEL GRADING

As has been previously stated, the amount of bond used in making a grinding wheel determines its hardness. The degrees of hardness are specified by use of the letters of the alphabet. The Norton Company and several other companies use a lettering system by which letters at the beginning of the alphabet indicate soft wheels and letters at the end of the alphabet indicate hard wheels. The lettering system for grade of bond is shown in an accompanying chart. The Carborundum Company uses a system in the reverse order. Another chart shows the bond classifications commonly used.

In the Norton system a wheel which is marked 3860-K5BE has the following characteristics: "38" represents the kind of abrasive, which in this case is ALUNDUM or Aluminous Oxide abrasive. "60" is the grain size, which according to the grain size chart is medium. "K" indicates the grade of the bond, which according to the chart for grade of bond is soft. "5" indicates the wheel structure; the numbers from 0 to 3 indicate close structure; 4 to 6 indicate medium structure; 7 to 12 indicate coarse structure. The last symbol in the wheel marking indicates the kind of bond. "BE" indicates the vitrified bond. "S" indicates a silicate bond; "T" is for the resinoid bond; "R" for the rubber bond; and "L" for the shellac bond.

CHART FOR GRADE OF BOND

Very Soft	Soft	Medium	Hard	Very Hard
E	H	L	P	T
F	I	M	Q	U
G	J	N	R	W
	K	O	S	Z

WHEEL SELECTION

There are several factors which affect the selection of a grinding wheel; they are the kind of material to be ground, the amount of stock to be removed, the accuracy as to size, the kind of finish required, the area of contact between the wheel and the work, and the kind of grinding machine to be used.

The nature of the material to be ground affects the selection of the wheel because, generally speaking, hard, dense materials require wheels possessing a soft bond with silicon carbide abrasive; materials that are soft and tough require a hard bond using aluminum oxide abrasive. The amount of material to be removed is important in selecting a grinding wheel because when there is a considerable amount of material to be removed the grains of a coarse grain wheel with wide spacing will be capable of taking a bigger, deeper cut without heating the work, but with slight sacrifice as to surface finish. When

COMMONLY USED GRADES AND GRAIN SIZES

	SURFACE	INTERNAL	EXTERNAL	CUTTER
GRADES	F G H I J K P	I J K L	J K L M P*	I J K L
GRAIN SIZES	36-46-60-80 120	34-46 60-120	46-60-80 120	36-46 60

*P for corners.

the amount of stock to be removed is slight, a wheel of fine grain and narrow spacing will take a smaller bite and give a good finish.

There are other factors which affect the grinding operation, such as: the speed of the wheel, the speed of the work, the condition of the grinding machine, and the knowledge and skill of the machine operator.

GRINDING THEORY

Grinding is the act of dressing, shaping, or finishing surfaces by means of a rotating abrasive wheel. In modern machine shop operation, it has been found by actual test that grinding costs will vary as much as 100% on the same work with the same kind of machine in the same factory. This is due to the difference in the handling and skill of operation of the machine by the mechanic. A good mechanic takes into consideration the factors involving the mounting, movement, size, and speed of the work and the mounting, movement, size, speed, and dressing or truing of the grinding wheel.

For precision grinding the work must be held rigidly to avoid vibration and to produce a good finish. If the work is held between centers, the center holes must be free from nicks, burrs, or dirt. The machine centers must be held securely and free from nicks. If held in a chuck or fixture, the work must be clamped so as to put the least strain on it and be solidly supported. After the work is correctly mounted the work speed must be selected so that it will move at approximately the right number of surface feet per minute to prevent distortion and excessive wear of the wheel face, and at the same time the traverse movement must be at a constant speed to prevent high and low spots on the work. The mechanism for moving the wheel must work smoothly and freely without play or bind to insure accuracy as to depth of cut.

The grinding wheel mounting is important because it must give steady and true motion to the wheel so that after it is trued up it will be free from vibration; it must give steady cutting action and be capable of producing a good finish after it is properly dressed.

GRINDING WHEEL SPEEDS

In most modern grinding machines the speed of the wheel is fixed as built by the manufacturer of the machine. Grinding wheels may be run from 5000 to 6000 surface feet per minute (S.F.P.M.). In case it is necessary to determine what the S.F.P.M. of a wheel is, place a tachometer or speed indicator in the center of the spindle and check the revolutions for one minute; then multiply the diameter of the wheel by 3.1416 which will give the circumference of the wheel in inches. The circumference multiplied by the R.P.M. will give the distance in inches which the wheel would travel in one minute if laid on its periphery and rolled at its given R.P.M. This result divided by 12 (12 inches to the foot) gives the surface speed in feet per minute.

EXAMPLE: Determine the surface speed in feet per minute of a grinding wheel 7" diameter mounted on a surface grinder. Using a tachometer the R.P.M. is found to read 3200. Multiply the diameter of the wheel by 3.1416 by 3200 and divide by 12:

$$\frac{7 \times 3.1416 \times 3200}{12} = \frac{70371.84}{12} = 5864.3$$

Thus the surface speed in feet per minute or the S.F.P.M. equals 5864.3

By being a good observer the operator can tell many things about the work and machine. Poor or defective bearings readily show up while the wheel is being dressed, for the diamond will not show a steady red spark as it moves across the face of the wheel.

CHATTER OR VIBRATION

If the work vibrates it will be shown by lines on the work parallel to the work axis, known as chatter. This may be remedied by changing to a softer wheel, tightening the spindle bearings, checking and repairing bad belt connections, taking a lighter cut or cutting down on the work speed. If the work shows mottled surfaces it is generally an indication of vibration due to irregular motion of the grinding wheel. This irregular motion may be caused by the lead core in the wheel being improperly placed, the wheel being water logged on one side, by the structure of the wheel being of uneven density, or the face of the wheel being improperly trued.

GLAZING AND LOADING

Glazing of the wheel is a condition in which the face or cutting edge takes on a glass-like appearance and is caused by the abrasive grains wearing away faster than the bond. As long as the bond is being worn

away as fast as the abrasive particles of the wheel are being dulled the wheel will continue to have good cutting action. To remedy this condition of glazing, use a wheel having a softer bond.

Frequently the material being ground clogs up or becomes imbedded in the pores of the wheel. This condition is known as "loading" or "pinning" and is caused by the work speed being too slow, the wheel action too hard, or crowding the wheel.

The continued use of the wheel after it becomes glazed or loaded puts an added strain on the work supports and wheel spindle bearings and forces the wheel out of true, and if continued too long will produce chatter marks or surface checks on the work.

DRESSING

The cutting face of a grinding wheel must be kept in a true, clean, sharp condition if the grinding operation is to be done efficiently. This requires frequent dressing or truing of the wheel. **DRESSING** is the operation of cleaning or fracturing the cutting surface of a wheel for the purpose of exposing new cutting particles. **TRUING** a wheel is the operation of removing material from the cutting face so that the resulting surface runs concentric with the wheel spindle axis.

The operations of truing and dressing grinding wheels are usually accomplished by using a commercial diamond or piece of tungsten carbide inserted in the conical point of a piece of cold rolled steel; by a diamond dust impregnated cement, formed into a stick and encased in metal tubing; or by a piece of silicon carbide mounted as a small wheel on an axle and placed in a cast iron base. In using any of these implements the point of the dresser is brought into contact with the face of the wheel by means of a special holder and then moved mechanically or by hand across the face of the wheel at a rate of speed which will produce the desired form or surface to the cutting edge.

A clean, true wheel of the proper bond and abrasive size is a very efficient cutting tool but at best will cause the work to heat up rapidly. In the case of the lathe tool there is only one cutting point acting on the work, but even so, it is a well known fact that the cutting tool, work, and chips get quite hot. In the case of a grinding wheel there are thousands of these cutting points each doing its share of the work but all acting at the same time, so that, since the action of a lathe tool generates heat, the action of a grinding wheel would necessarily develop a much greater heat. For that reason a flood of lubricant coolant at the point of contact between the wheel and work is necessary to carry off the heat and to keep the temperature of both the wheel and work as nearly constant as possible. This is especially true where a job is roughed and finish ground.

ROUGH GRINDING

The purpose of roughing the job is to remove internal strains set up by heat treatment and to remove the excess stock as rapidly as possible in preparation for finishing. Roughing is accomplished by using a slow work speed and a fast traverse, then when finishing use a high work speed and slow traverse. In both cases use a flood of good coolant.

The following are suggestions originally made by well known grinding wheel and machine manufacturers and if generally adopted should do much to eliminate grinding accidents.

CARE OF GRINDING WHEELS

1. Handle all wheels with the greatest care in storing or delivery. Wheels are frequently cracked by rough usage long before they are ever placed on a grinding machine.
2. Wheels should be stored in a dry place.
3. Before a wheel is placed on the spindle it should be sounded for cracks. A solid wheel, when tapped by a non-metallic object, gives off a dull ringing sound. A cracked wheel gives off a dull thudding sound.
4. Make sure that the grinding wheel is equipped with blotting paper gaskets on each side.
5. Never crowd a wheel on the spindle; the hole in the wheel should be .003 to .005 inch oversize to permit it to slide easily on the spindle and squarely against the flange.
6. Never mount a wheel without flanges which are properly relieved and of suitable proportions. (See Wheel Mounting, page 236.)
7. Don't screw the wheel nut too tight. The nut should be set up only tight enough so that the flanges hold the wheel firmly.
8. Keep the wheel clean and true by frequent dressing, but don't remove any more stock than is necessary to put the wheel in proper condition.
9. If a wheel vibrates excessively after it

has been properly trued, there is something wrong. Stop the machine and call an instructor.

10. Large wheels (that is wheels over 12") require special balancing. Don't attempt to balance them yourself.

Revolutions per Minute for Various Diameters of Grinding Wheels to Give Peripheral Speed in Feet per Minute as Indicated

Diameter of wheel in inches	Peripheral Speed in Feet per Minute											
	4000	4500	5000	5500	6000	6500	7000	7500	8000	8500	9000	9500
	Revolutions per Minute											
1	15279	17189	19098	21008	22918	24828	26737	28647	30558	32467	34377	36287
2	7639	8594	9549	10504	11459	12414	13368	14328	15279	16233	17188	18143
3	5093	5729	6366	7003	7639	8276	8913	9549	10186	10822	11459	12115
4	3820	4297	4775	5252	5729	6207	6685	7162	7640	8116	8595	9072
5	3056	3438	3820	4202	4584	4966	5348	5730	6112	6494	6876	7258
6	2546	2865	3183	3501	3820	4138	4456	4775	5092	5411	5729	6048
7	2183	2455	2728	3001	3274	3547	3820	4092	4366	4538	4911	5183
8	1910	2148	2387	2626	2865	3103	3342	3580	3820	4058	4297	4535
10	1528	1719	1910	2101	2292	2483	2674	2865	3056	3247	3438	3629
12	1273	1432	1591	1751	1910	2069	2228	2386	2546	2705	2864	3023
14	1091	1228	1364	1500	1637	1773	1910	2046	2182	2319	2455	2592
16	955	1074	1194	1313	1432	1552	1672	1791	1910	2029	2149	2268
18	849	955	1061	1167	1273	1379	1485	1591	1698	1803	1910	2016
20	764	859	955	1050	1146	1241	1337	1432	1528	1623	1719	1814
22	694	781	868	955	1042	1128	1215	1302	1388	1476	1562	1649
24	637	716	796	875	955	1034	1115	1194	1274	1353	1433	1512
26	588	661	734	808	881	955	1028	1101	1176	1248	1322	1395
28	546	614	682	750	818	887	955	1023	1092	1159	1228	1296
30	509	573	637	700	764	828	891	955	1018	1082	1146	1210
32	477	537	597	656	716	776	836	895	954	1014	1074	1134
34	449	505	562	618	674	730	786	843	898	955	1011	1067
36	424	477	530	583	637	690	742	795	848	902	954	1007

To find the R.P.M. of an 18-inch wheel having a surface speed of 6000 F.P.M., read across in the 18 line to the 6000 column which gives 1273 R.P.M.

1. What is meant by the term abrasive?
2. Name two natural abrasives.
3. Name two artificial abrasives.
4. Why are artificial abrasives used more than natural abrasives?
5. What are the characteristics of silicon carbide?
6. Name four materials on which silicon carbide should be used.
7. On what kinds of material should aluminous oxide be used?
8. Why is control of the abrasive grain size important?
9. How is the abrasive grain size determined?
10. How are the abrasive particles formed into a wheel?
11. What determines the hardness or softness of a wheel?
12. Name five commonly used bonds.
13. Which bond is used most frequently? Give five reasons.
14. What are the advantages of the silicate bond?
15. Silicate bonded wheels are generally used for what purpose?
16. Rubber bonded wheels are used for what purpose?
17. How does the wheel structure affect the grinding operation?

18. How are the degrees of hardness of the wheels specified?
19. What is the difference between the Norton and Carborundum methods of rating hardness.
20. Into what five groups are the degrees of hardness divided?
21. What six factors govern the wheel selection for a given job?
22. Give four methods of overcoming "chatter."
23. What is the difference between "glazing" and "loading?"
24. What is the difference between "dressing" and "truing?"
25. Why is it necessary to keep the face of the wheel clean and true?
26. For general tool room work which abrasive is most generally used and why?
27. How might the grade and structure of the grinding wheel produce chatter?

Chapter 25

GRINDING MACHINES

DEFINITION

A grinding machine is a machine which employs a grinding wheel for producing cylindrical, conical, or plane surfaces accurately and economically and to the proper shape, size, and finish. The surplus stock is removed by feeding the work against the revolving wheel or by forcing the revolving wheel against the work.

GRINDING MACHINE TYPES

There is a great variety of grinding machines, but it is the purpose of this text to treat only of those machines which are regularly used in tool and die shops. The machines which are generally used are the cutter grinder, surface grinder, centerless grinder, external grinder, internal grinder, tool grinder, and flexible shaft or hand grinder. These machines are usually classified as to size by the largest piece of work which they can completely machine (as 6" X 18" External Grinder) or are specified by numbers such as Brown and Sharpe #13, Rivett #104 Internal, and #3 Abrasive Surface Grinder.

CUTTER GRINDER

A cutter grinder is a machine that holds the cutter while a rotating abrasive wheel is applied to the edges for the purpose of sharpening it. These machines vary in design from simple machines having a limited purpose to complex universal machines that can be adapted to any cutter grinding requirements. Fig. 723 is an illustration of a typical cutter grinder used in the Henry Ford Trade School.

The universal cutter grinder is capable of grinding cutters of various shapes by using special attachments and specially formed grinding wheels. Generally the flaring cup, plain, and dish or saucer wheels are used on cutter grinders (Fig. 722).

In selecting a grinding wheel for general purpose cutter grinding, select a soft, free cutting wheel and take very light cuts so that the temper is not drawn from the edge of the teeth. Generally speaking, wheels of grain size from 30 to 60 and J or K bond are best adapted for high speed cutters. The shape of the wheel depends on the shape of the cutter to be sharpened.

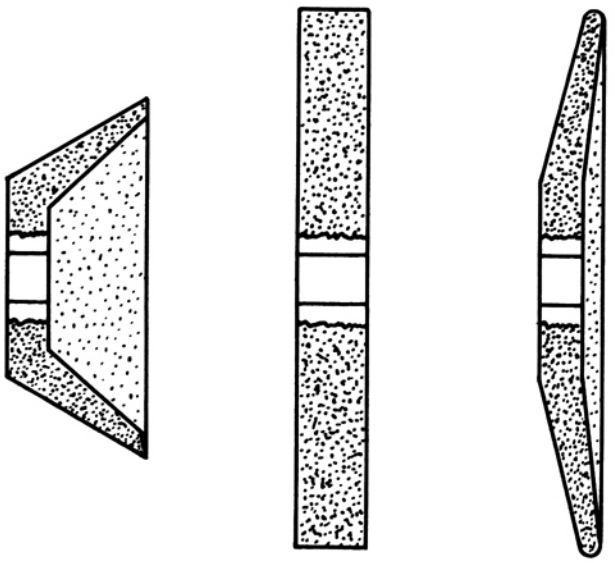


Fig. 722

SHARPENING OF CUTTERS

To sharpen the teeth of a plain milling cutter, the cutter is generally mounted on a lathe mandrel and supported between centers (as shown in Fig. 724) or the cutter is mounted on a special stub arbor and held in a universal swivel fixture. After the cutter has been mounted in the machine, a tooth rest is mounted on the table or work head and adjusted to the tooth to be first sharpened. The table and tooth rest are adjusted so that the grinding wheel follows the original land on the back of the tooth and gives the proper clearance. The cutter is then fed to the rotating grinding wheel until sparks indicate contact between it and the wheel; then the table is moved back and forth traversing the cutter until the wheel is finished cutting. Revolve the cutter backwards 180° against the spring tension of the tooth rest and without changing the depth of cut take a trial cut on this opposite tooth to check for taper. If no taper is apparent, revolve the cutter backwards and engage the next tooth to be

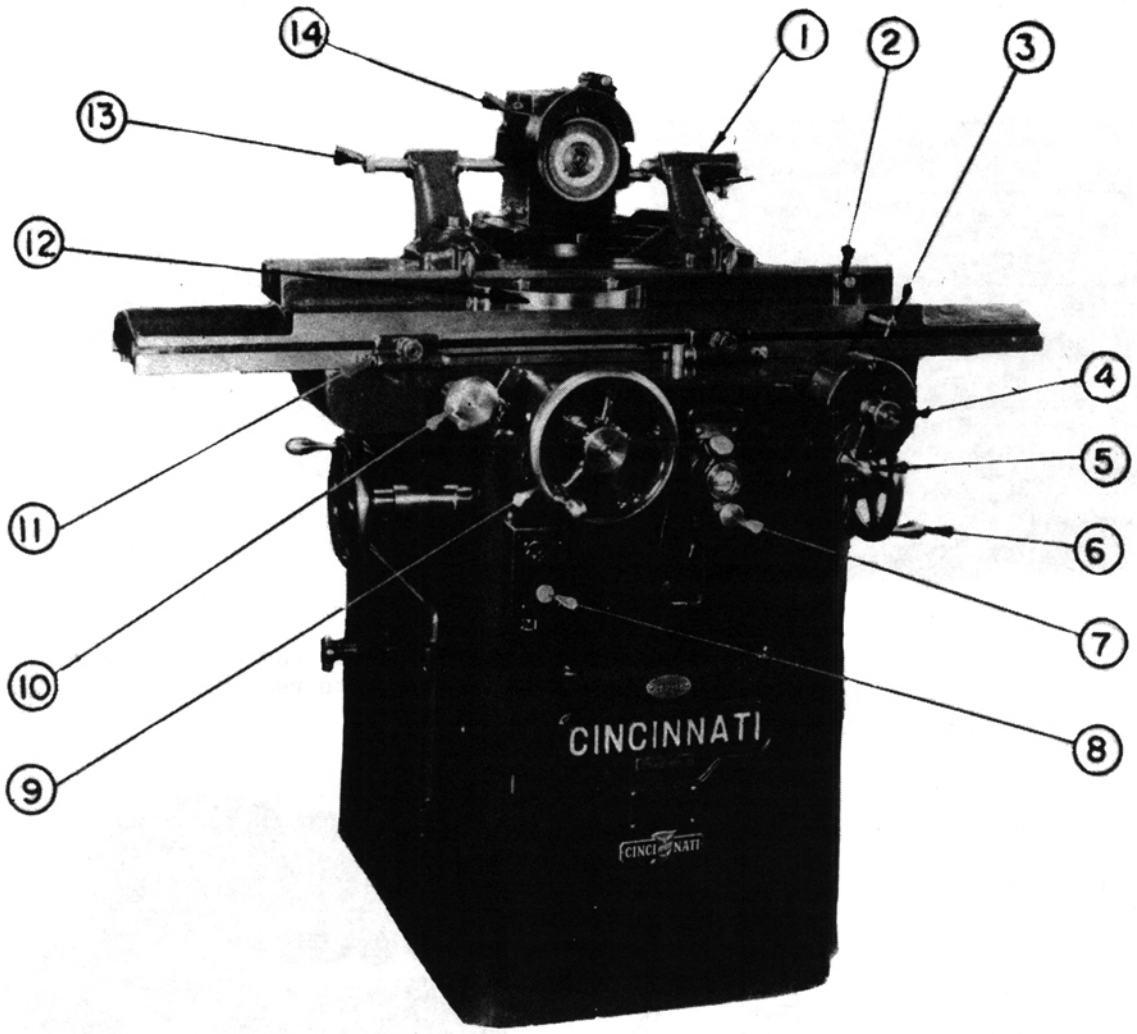


Fig. 723. Universal Cutter Grinder.

1. Quick acting tail stock.
2. Table swivel lock.
3. Fine adjustment for table swivel.
4. Slow hand movement for traversing table.
5. Crank for fast table traverse.
6. Handwheel for elevating grinding wheel head (one on each side).
7. One shot lubricating system.
8. Electrical control buttons.
9. Handwheel for adjusting table saddle (one front and rear).
10. Handwheel for fast movement of table traverse.
11. Adjustable cushioning dogs.
12. Graduated dial for swiveling table.
13. Work head center.
14. Wheel head can be raised or lowered and swiveled.

sharpened. This process is repeated until all teeth have been sharpened and ground concentric.

TOOTH RESTS

Tooth rests are of two kinds, plain and hook (Fig. 725). They consist of a piece of spring steel about .030" thick, 1/2" to 1-1/2" wide, and from 1" to 3" long brazed or riveted in a piece of round 3/8" cold rolled steel. They are held by means of a forged clamping fixture (Fig. 726) by bolting the fixture either to the work table or the grinding wheel head.

SHARPENING SIDE TEETH OF MILLING CUTTER

The peripheral teeth of a side milling cutter are sharpened the same way as those of a plain milling cutter. To sharpen the side teeth the cutter is generally mounted on a stub expansion arbor and placed in the universal swivel fixture (Fig. 727). The

swivel fixture is then adjusted so that the side teeth will be ground about 1° out of

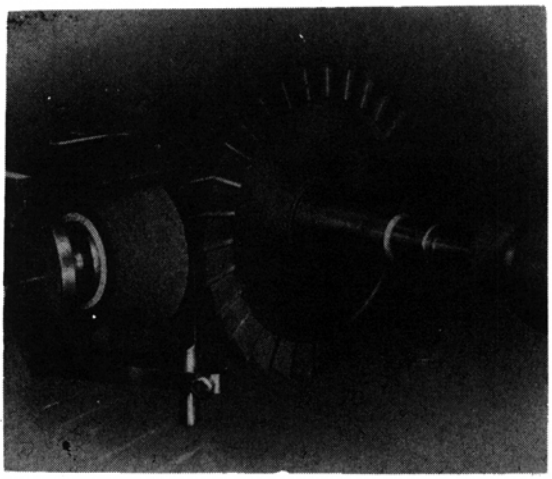
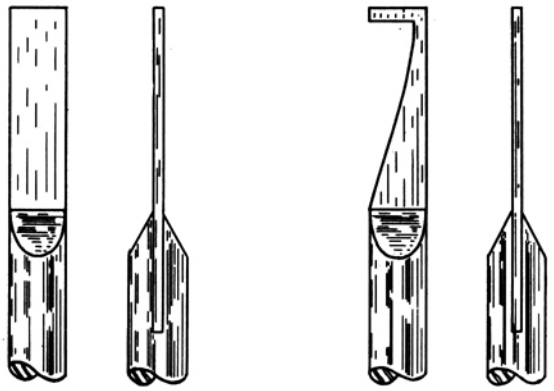


Fig. 724

parallel with the side of the cutter and have a side clearance angle of from 3° to 5°. The



(a) (b)

Fig. 725. (a) Plain; (b) hook.

general practice is to give the teeth of roughing cutters a side clearance of 5° and finishing cutters 3°. It is important in all cutter grinding to grind all teeth to the same height so that each tooth will do its share of cutting. For that reason after the teeth have all been sharpened a very light cut is taken on each tooth to insure them all being the same height.

SLABBING CUTTERS, that is, cutters which are of considerable length relative to their diameters, if they have straight teeth are sharpened the same way as a plain milling cutter.

HELICAL TOOTH CUTTERS are ground and mounted the same as other cutters, but the tooth rest must be mounted in a fixed posi-

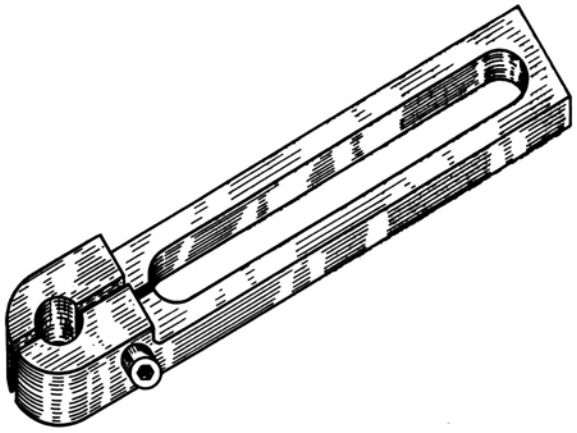


Fig. 726

tion relative to the grinding wheel. Generally the tooth rest is mounted on the

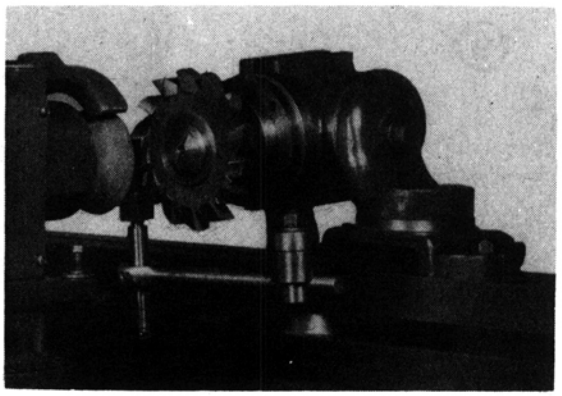


Fig. 727

wheel head (as shown in Fig. 724). This is done so that the cutter is forced to slide over the tooth rest causing the cutter to turn in such a manner that the tooth being ground will have the same helical shape at which it was originally milled. When the cutter and tooth rest are properly mounted and the grinding ready to start, the tooth should be pressed lightly against the tooth rest and held there while the table is moved longitudinally. If this is not done the tooth will leave the rest and the cutter will be damaged (see Fig. 728).

FORMED CUTTERS are sharpened by grinding the face of the teeth radially, that is by grinding the face of the teeth with a dish or saucer wheel so that the face

of the tooth comes on the radius of the cutter. (See Fig. 729.)

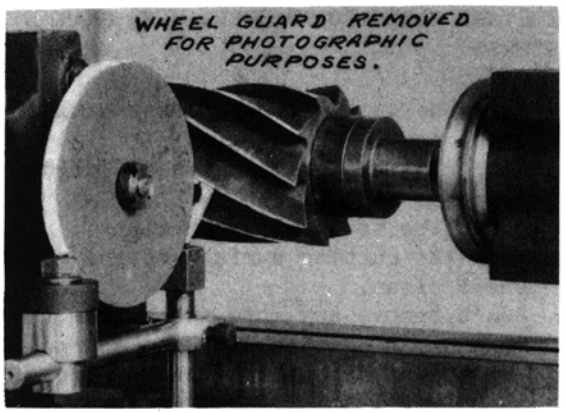


Fig. 728

The machine is set up for grinding the formed cutter by bringing the centers in line with the face of the grinding wheel (Figs. 729, 730 and 731). If this is not done the cutter will not have the correct shape. The cutter, mounted on a mandrel is placed between the centers and the face of the tooth brought against the grinding wheel. The tooth rest is then set against the back of the tooth (Fig. 731). Move the table longitudinally and clear the wheel from the work, start the wheel and take a trial cut. To adjust the work to the wheel while grinding, revolve the cutter by moving the tooth rest toward the grinding wheel. This practice keeps the faces of the teeth radial and maintains the correct shape of the tooth.

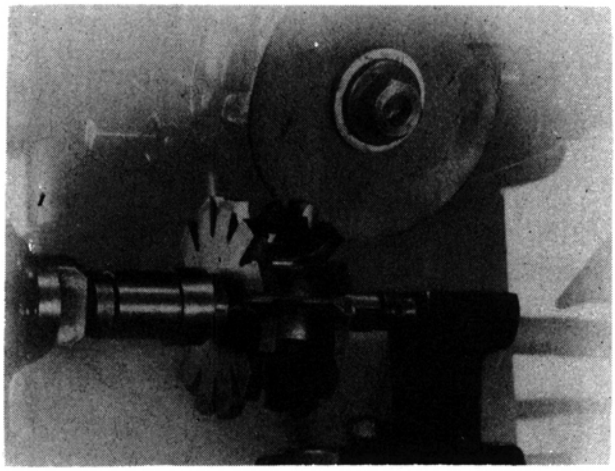
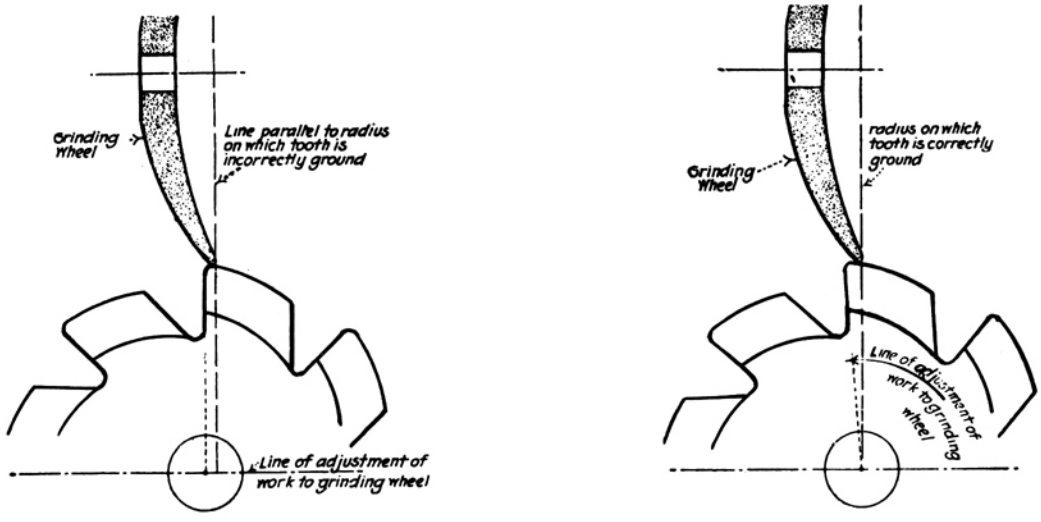


Fig. 730

TO SHARPEN ANGULAR CUTTERS, the cutter is mounted on a stub expansion arbor and placed in the swivel fixture (Fig. 732).

The fixture is then swiveled to the desired clearance angle and the tooth rest set on the exact centerline of the cutter. Adjust the grinding wheel to the cutter by raising or lowering the table so it will grind the tooth supported by the tooth rest and allow the tooth immediately above to clear the grinding wheel as illustrated. The tooth rest should never be set above or below center as this will cause a change in the actual angle ground on the cutter from the


Grinding wheel
Line parallel to radius on which tooth is incorrectly ground
Line of adjustment of work to grinding wheel
Grinding wheel
radius on which tooth is correctly ground
Line of adjustment of work to grinding wheel

(a) (b)

Fig. 729. (a) The Wrong Way to Grind a Gear Cutter. This brings the faces of the teeth parallel to but behind the radius and spoils the tooth outline, and therefore makes the cutter inaccurate. (b) The Correct Way to Grind a Gear Cutter. The faces of the teeth will always be radial.

one at which the machine has been set and is often the cause of angular cutters not being accurate.

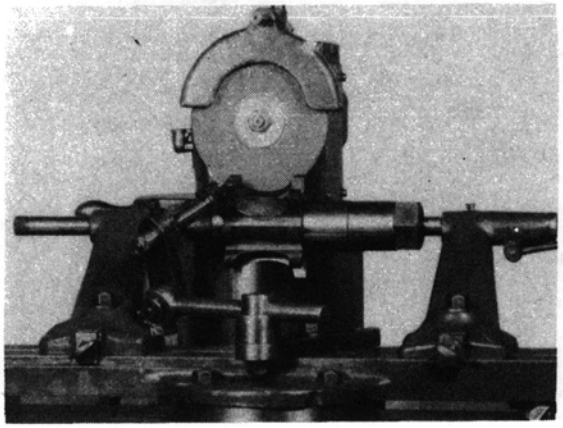


Fig. 731

END MILLS, whether they are straight or helical, are ground the same as milling

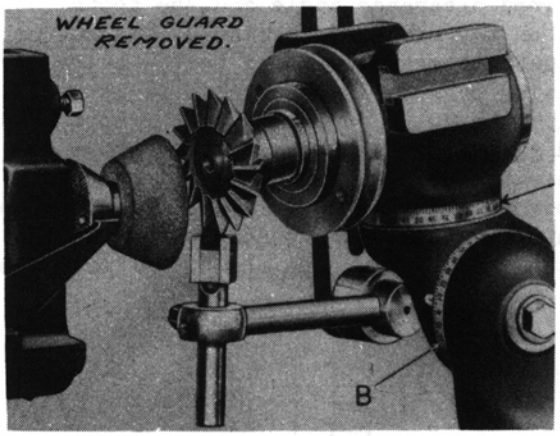


Fig. 732

cutters. The end teeth are ground the same as the side teeth of a side milling cutter.

As was previously mentioned there are three wheel shapes commonly used for cutter grinding. They are the DISK, FLARING CUP, and SAUCER wheel. The saucer wheel is used for grinding formed cutters; the disk wheel is used for grinding the clearance on cutters having narrow lands on the teeth; the flaring cup wheel is used for grinding clearance on teeth having wide lands and for gumming out slitting saws, and the spacing between the teeth of milling cutters.

In actual practice the grinding wheel revolves downward toward the cutting edge so that the action of the wheel forces the tooth against the tooth rest. This practice results in a burr or wire edge being left on the tooth which should be oil-stoned off.

IMPORTANT FACTS TO REMEMBER WHEN SHARPENING CUTTERS:

1. Keep the cutter tooth firmly against the tooth rest.
2. Make sure the grinding wheel follows the original land on the tooth.
3. Mount the tooth rest correctly.
4. Keep the cutting surface of the grinding wheel clean.
5. Don't remove any more stock from the tooth than that required to sharpen it.
6. Be careful not to draw the temper of the tooth.

The following SAFETY RULES for cutter grinders must be observed at all times:

1. Always wear goggles on all cutter grinder work.
2. Under no circumstances is the machine to be started unless the grinding wheel is adequately guarded. Use a guard of the proper size and adjust it closely to the wheel, allowing the minimum amount of wheel exposure with which to work.
3. In mounting wheels on cutter grinders, use standard wheel bushings and safety washers. Use paper washers on large wheels.
4. When hand dressing wheels be careful to allow ample hand clearance between the wheel and table or other parts of the machine.
5. Hand dressing operations should be performed with a light pressure, especially when dressing thin wheels. A slip of the hand or a broken wheel may cause severe lacerations.
6. Any changes of guards, dogs, centers, setup, tooth rests, or any other parts of a machine are not to be made while the machine is running.
7. When grinding spotfacers, counterbores, etc., on a draw collet, use a special automatic safety guard or shut the machine down to remove the work.
8. In backing off drills, spiral reamers, etc., see that the tooth rest is properly adjusted in relation to the wheel and work to prevent slippage and consequent spinning of stock. Ask the instructor about this.
9. Care should be taken in handling sharp tools, such as reamers, drills, cutters,

and counterbores, as severe lacerations may result from stock slipping through the hands.

10. Towels are not to be used in holding small tools such as spotfacers, counterbores, and similar tools that become warm while grinding. Ask the instructor how to take care of work of this class.
11. Exhaust hoods are supplied as a safeguard for the health of grinder operators. See that they are properly adjusted at all times and that they are not abused.

QUESTIONS

1. What is a cutter grinder?
2. Name three manufacturers of universal cutter grinding machines used in the school.
3. What kinds of cutters can be sharpened on the universal cutter grinding machine?
4. What kinds of grinding wheels are suitable for general purpose cutter grinding?
5. What is a tooth rest and why is it used in cutter grinding?
6. How are plain milling cutters held for sharpening?
7. After the work and tooth rests are correctly mounted, how is the tooth sharpened?
8. Name three shapes of wheels used for cutter grinding.
9. How are the teeth of side milling cutters sharpened?
10. How are slabbing cutters having helical teeth sharpened?
11. What caution should be exercised in sharpening the teeth of helical tooth cutters?
12. How are the teeth of formed tooth cutters ground?
13. Why are formed tooth cutters ground radially on the face?
14. What is the shape of the wheel used for formed tooth grinding?
15. How are end mills generally ground?
16. Why is it important to prevent burning of the cutting edge in cutter grinding?
17. What causes burning of the cutting edge in cutter grinding?
18. Name the two types of tooth rests most generally used?

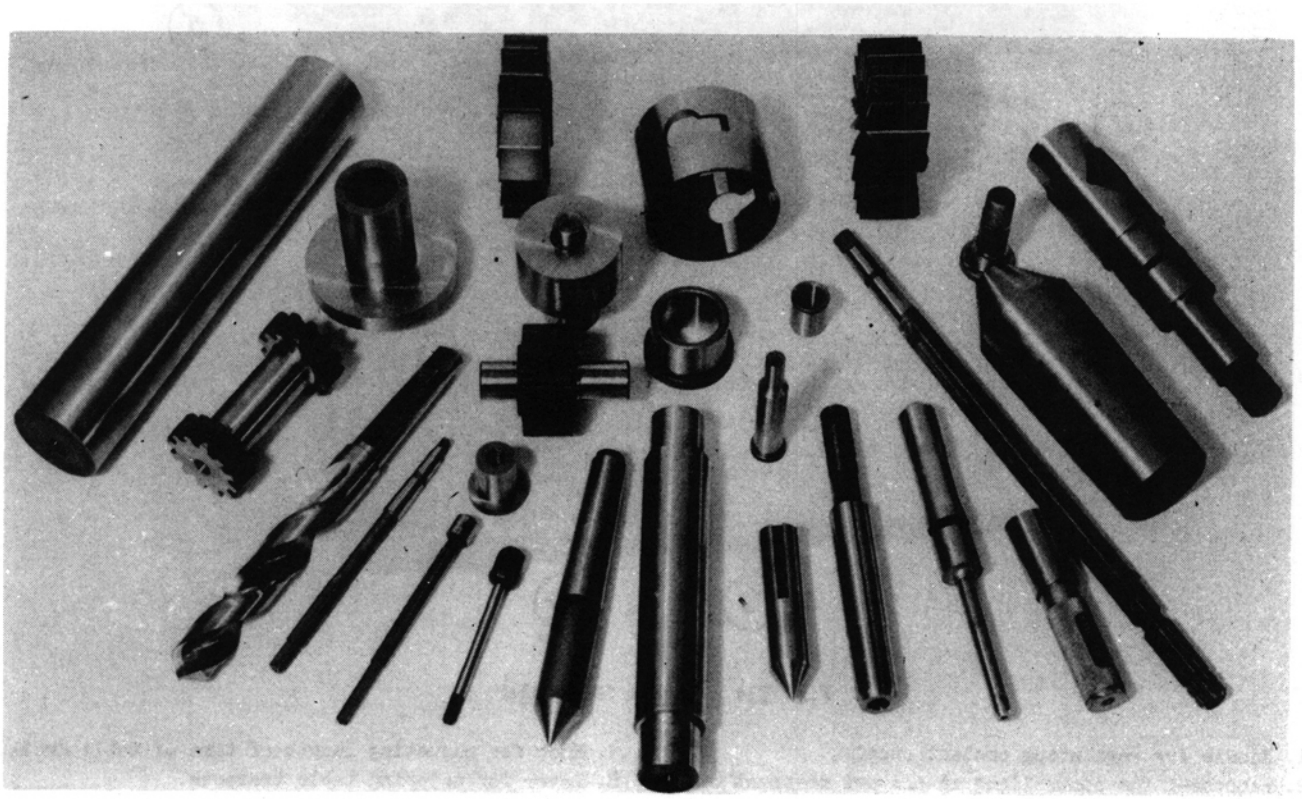


Fig. 733. Samples of external grinding as done in the Henry Ford Trade School.

CYLINDRICAL GRINDING (Shop term - External, OD; Internal, ID)

External grinding, as commonly thought of, is the act of grinding the outside diameter of a piece of work while it is revolving on its axis, for the purpose of reducing it to size, leaving a fine finish. However, external grinders are used to produce external cams, eccentrics, and special forms on the outside diameter of work. They are machines which are capable of doing many of the operations done on a lathe, but much more accurately. The big advantage lies in the fact that after a job has been heat treated a good surface finish and extreme accuracy as to size can be obtained and where these two factors are important the extra cost involved in grinding can be overlooked. If parts can be designed so that the amount of stock to be removed is within grinding limits, then grinding is much less costly than lathe turning.

External grinders are divided into three general groups. They are the PLAIN CYLINDRICAL, UNIVERSAL, and special grinders like the CENTERLESS and CAM grinders.

The PLAIN CYLINDRICAL grinder (Fig. 734) is used to produce external cylinders, tapers, fillets, undercuts, shoulders, and may be used for form grinding by dressing the desired contour on the grinding wheel.

In any cylindrical grinder there are three movements which are very important.

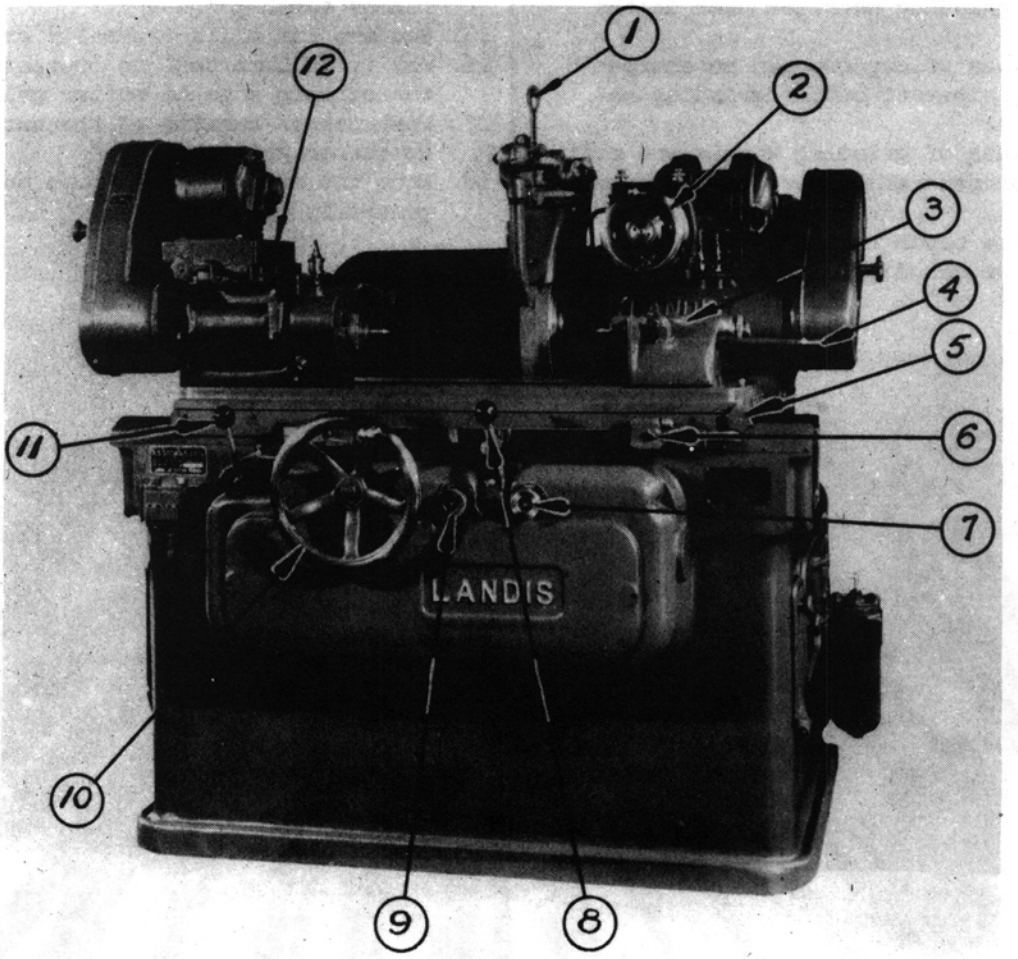


Fig. 734. Landis OD Grinder

1. Handle for regulating coolant supply.
2. Handwheel for controlling in-and-out movement of grindwheel.
3. Footstock. Adjustable on table.
4. Lever for controlling movement of footstock center.
5. Knob for adjusting table for taper.
6. Trip dogs. Can be adjusted for length of work.
7. Knob for adjusting length of time of table dwell.
8. Lever for tripping table traverse.
9. Knob for regulating speed of table traverse.
10. Handwheel for traversing table by hand.
11. Lever for starting and stopping headstock spindle and table traverse.
12. Headstock unit.

They are rotation of the work on its axis, movement of the work back and forth in front of the wheel, and movement of the wheel into the work. Trouble in cylindrical grinding, with exception of wheel content or make up, can be attributed to one of the three movements mentioned.

Rotation of the work on its axis is important because if the centers in the work are bad, or if the machine centers are of poor quality and loose, the work will be irregular in form. The movement of the work back and forth in front of the wheel must be steady and smooth to insure a good finish and accurate sizing of the work. Movement of the wheel spindle to revolve the wheel must be true and smooth to prevent vibration and to avoid chatter marks on the work. Movement of the wheel into the work must be without play or bind to insure accuracy as to depth of cut.

The work in external grinding usually revolves on two dead centers, one in the foot stock and one in the head stock, and is given its rotary motion by means of a drive plate, which revolves about the head stock center, driven by a motor using pulleys and a belt (Fig. 735). The drive plate contains an adjustable arm which can be located at varying distances from the center and into which a drive pin is fitted. This drive pin engages in the "V" slot of the grinder dog which is attached to the work and hence the revolving motion of the plate is transmitted to the work.

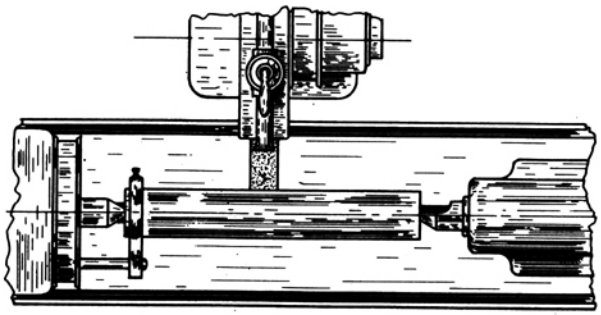
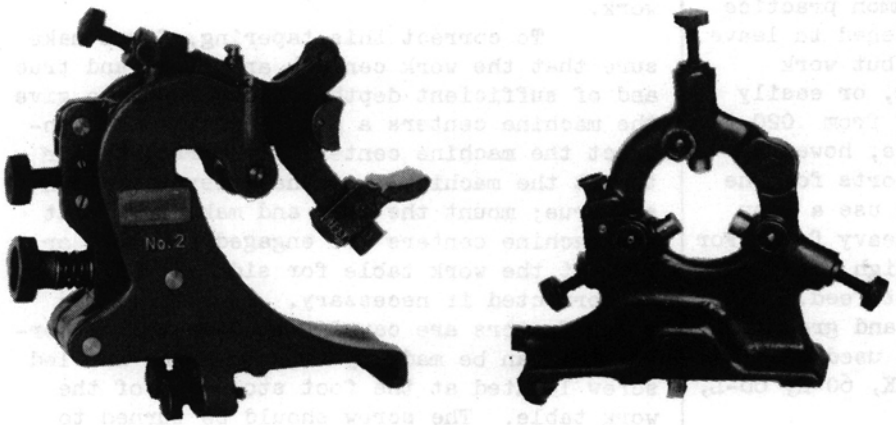


Fig. 735

Besides the rotary motion of the work it is always necessary, except in the case of "plunge cut" grinding, for the work to be traversed past the grinding wheel. The length of the table traverse should be set so as to permit the wheel to run off the end of the work about one-third of the wheel face width. If the wheel is not permitted to over-run the end of the work, the job will be oversize at that point because the wheel will not have a chance to finish the cut. If the wheel is permitted to completely over-run the end of the work, the job will be undersize, because the pressure required between the wheel and the work is relaxed thus permitting the work to spring toward the wheel, and at the beginning of the traverse the wheel would cut undersize.

At the end of each traverse the table stops momentarily to give the wheel a chance to grind the work to size, to permit the wheel to clear itself on the new cut and to avoid the jarring motion which would be unavoidable with an immediate reversal of the table. If this traversing is not accomplished without jarring or without a jerking movement of the table, the work will show high and low spots due to the slight pause or dwell of the wheel on the work, which would take place at each jerk or jar of the table. The speed of the traverse depends on the width of the wheel face and the finish required on the work. It is generally such that the table will move two-thirds to three-fourths of the wheel face for each revolution of the work.



(a) (b)

Fig. 736

STEADY REST

Long slender work, besides being supported by the centers, should also be supported with steady rests. Fig. 736(a) shows a common type of steady rest while Fig. 736(b) shows a center rest.

When using steady rests (Fig. 737) it is very

important to keep the jaws properly adjusted to the work, otherwise the work might get caught between the lower jaw and the wheel and be thrown from the machine or break the wheel.

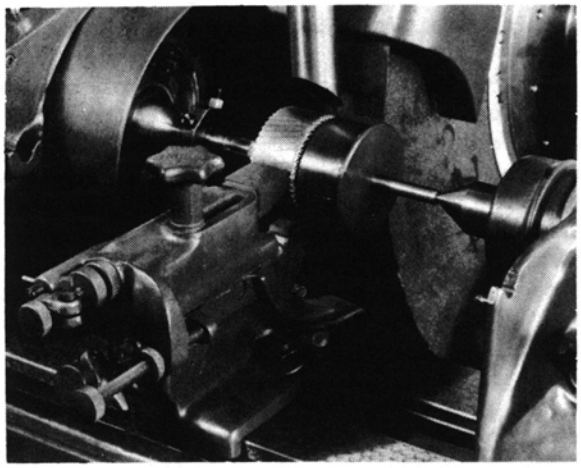


Fig. 737

FEEDS

The grinding wheel can be fed to the work either by automatic or hand feed, and permits feeds as small as .00005. It is not advisable to use the hand feed except to bring the wheel up to the work or move it away or when taking very fine cuts. The automatic feed takes cuts from .00025 to .004 for each traverse of the table and saves time and wear and tear on the machine by taking more uniform cuts and gives longer life to the grinding wheel. Generally speaking, "roughing cuts" may be from .001 to .004 at each reversal, depending on the rigidity of the machine and work set up and the amount of stock to be removed. It is common practice when the work is not to be hardened to leave from .006 to .010 grind stock, but work which is large, long or slender, or easily sprung may have an allowance of from .020 to .030 grind stock; in either case, however, heavy infeeds require good supports for the work. For roughing out the job use a slow work speed, fast traverse and heavy feed. For "finish grinding" a job use a high work speed, slow traverse and a light feed.

The more common grades and grain sizes of aluminous oxide wheels used for external grinding are: 46-J, 46-K, 60-K, 60-L, 80-O, 120-P.

WORK SPEEDS

To the question, "How fast should the work speed be?" no set rule can be given. Jobs may or may not be of the same metal, same heat-treatment, or same diameter, any one of which factors affects the speed. Too fast a speed tends to wear away the wheel faster, while too slow a speed causes the wheel to cut harder and become dull and glazed. If the wheel wears too fast it requires more frequent dressing and truing and takes more time to do a given job. Extremely hard steels may have to be ground at 30 S.F.P.M., while very soft steels may require up to 100 S.F.P.M. for finishing. Common work speeds are from 30 to 50 S.F.P.M. If the wheel becomes dull or glazed it will burn the work because of the added friction set up by forcing the wheel to cut. If the work speed is correct and the wheel still has a tendency to become glazed it is an indication that the bond is too hard and can be corrected by substituting a wheel of the same grain but softer bond.

OPERATION ERRORS

When cracks, checks, or burns show up on the surface of the work any one of a combination of the following conditions might be the cause: improper speeds of work, wheel too hard, or the wheel being glazed or loaded. If the wheel is glazed or loaded it will cause the work to burn and crack even though there is a good supply of lubricant, so that best results can be obtained by keeping the wheel sharp and clean at all times.

Perhaps the greatest obstacle which the operator has to overcome in the grinding of a cylinder is that of taper in the work. Conditions which contribute to it are poor machine or work centers, work improperly mounted, work table out of adjustment, foot stock out of line with head stock, or steady rests bearing too heavily or out of line with the work.

To correct this tapering, first make sure that the work centers are clean and true and of sufficient depth and clearance to give the machine centers a good bearing; then inspect the machine centers for correct taper to fit the machine; see that they are clean and true; mount the work and make sure that the machine centers are engaged properly or inspect the work table for side play and have it corrected if necessary. If none of the above factors are causing the taper then correction can be made by adjusting the knurled screw located at the foot stock end of the work table. The screw should be turned to move the small end of the work away from the wheel. Several adjustments may be necessary to get the work entirely cylindrical.

PLUNGE CUT GRINDING

"Plunge cut" grinding is cylindrical grinding where the length of the work is less than the width of the wheel face so that the wheel can be fed straight into the work and completely grind the cylinder without using the table traverse. When it is necessary to grind a shouldered or stepped cylinder, plunge cut grinding is used to maintain a square corner between the cylinder and the shoulder.

The important steps in external grinding a piece of work are as follows:

1. Check the work for size to make sure grinding stock has been allowed and at the same time note any tapering of the work
2. Inspect the work centers for being clean and true. Select machine centers of suitable diameters to properly fill the work centers. The foot stock center should be cut away enough to permit the grinding wheel to clear the end of the work; a center of this type is known as a one-half full, or three-quarter full center.
3. Attach the grinding dog on the end of the job making sure that the dog does not damage such parts of the work as threads, keyways, etc., then lubricate the machine centers.
4. Set the table traverse for the length of the work, allowing for overrun of the end and the space that the grinder dog occupies.
5. If necessary, mount the steady rests and adjust the shoes to the work.
6. Dress the grinding wheel, passing the diamond across the face of the wheel quite fast, to make the wheel fast cutting. Set the work speed at the correct S.F.P.M.
7. Feed the wheel to the work by hand and take a light cut noting that the wheel starts to cut approximately at the high point of the work to conform to the check in step one.
8. Check the work for size and taper and make any table adjustments necessary to insure that the work will be straight.
9. Rough grind the job to rough size. If several pieces are to be done, set the stop on the feed ratchet and proceed as before, roughing the balance of the pieces.
10. After the pieces have been roughed out in this manner, place the dog on the rough ground end of the work and grind the unfinished end. If this end is shorter than the width of the wheel face it may be plunge cut ground. Make sure that the grinding wheel is kept sharp and clean by frequent dressing.
11. To finish grind, set the machine for fast work speed and slow traverse and dress the wheel by passing the diamond slowly across the wheel face.
12. Insert the piece to be finish ground and take a light trial cut. Check it for size and make any corrections necessary for removal of taper and if steady rests are used keep them adjusted to the work.
13. After the first piece has been ground to finish size reset the stop on the feed ratchet so that the infeed will produce the required size and then set the shoes on the steady rests for the finished diameter.

The above outline pertains to grinding a plain cylinder. If the work to be ground has shoulders, keyways, or slots, some deviations from the outline must be made. If the work to be ground has a keyway, open at each end, or splines and steady rests are to be used, it should be filled with key stock or other suitable material to prevent the steady rest shoes from catching on the work.

Sometimes it is desirable to finish grind a cylindrical piece in one operation. When a condition of this kind arises, an angle iron bracket or other suitable projection can be sweated to the end of the work (Fig. 738) to act as a driver, but it must be fixed so as not to interfere with the overrun of the grinding wheel or work center.

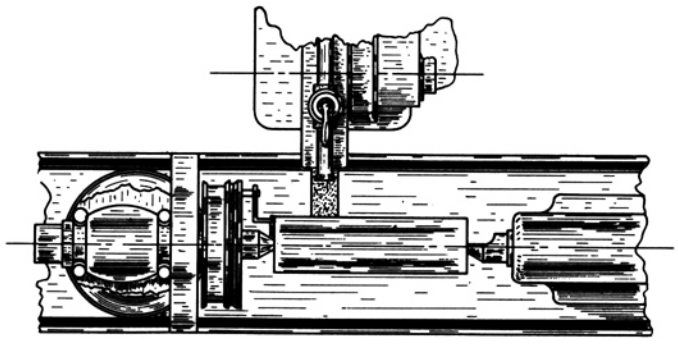


Fig. 738

SHOULDER WORK

If the work must be ground to a shoulder, it is advisable to locate the grinding wheel up against the shoulder and by plunge

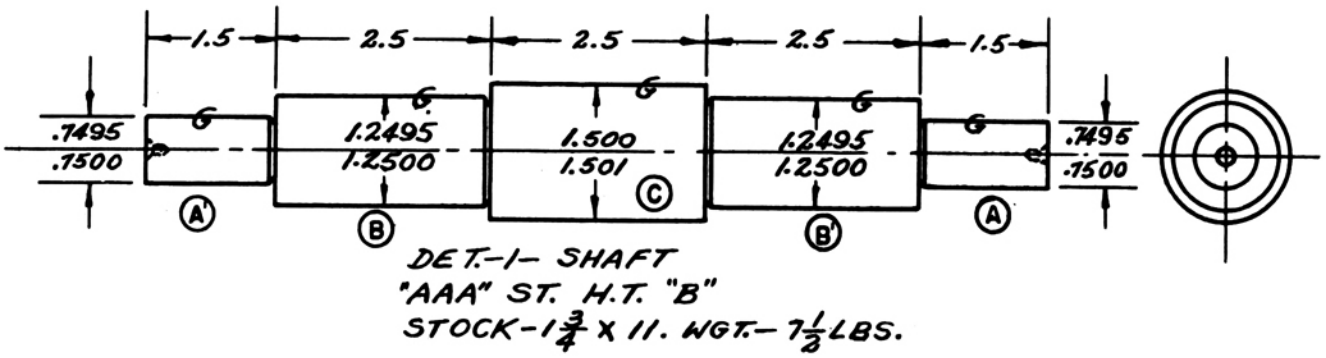


Fig. 739

cut grinding, grind the work to the correct diameter. This method leaves the finished diameter with a square corner at the shoulder rather than a filleted one which would be left if the work was traversed up to the shoulder. After the job has been sized square with the shoulder, the balance of it can be done by traversing the table.

THE FOLLOWING OUTLINE IS A TYPICAL ANALYSIS OF THE PROCEDURE IN EXTERNAL GRINDING:

Type of job	Shaft as per sketch. (Fig. 739)
Type of machine	External grinder
Type of steel	"AAA"
Heat treatment	"B"
Kind of grinding wheel.	60-L
Operations required . .	Rough and finish grind as per sketch.

PROCEDURE

1. Check all diameters for sufficient grind stock.
2. Check work centers for being free from dirt and nicks.
3. Dress grinding wheel for roughing cut.
4. Mount a grinder dog of correct size on the work.
5. Set foot stock to function for correct length of work.
6. Mount the work in the machine and properly adjust the drive pin to the dog.
7. Feed the grinding wheel to diameter "A" and take a clean-up cut, seeing that the wheel closely follows the work.
8. Check diameter "A" for straightness, make any necessary table adjustments, take a trial cut and recheck the work for straightness.
9. With the wheel cutting straight, plunge cut diameter "A" at the shoulder and rough grind, leaving .003 to .005 for finishing. Repeat this operation on A', B, B', and C in the order stated, then have the job inspected.
10. Dress the grinding wheel for finishing and with the machine cutting straight, finish grind A, A', B, B', and C in the order stated.
11. Have job inspected on all dimensions.

GAGING METHODS

CYLINDRICAL WORK having dimensions which must be held to close limits requires the use of gages which are calibrated to a finer degree of accuracy than ordinary micrometers. For this purpose an "amplifying comparator" is used. These amplifiers are of several types and calibrations, but the kind commonly used has a ratio of 10 to 1. This means that one division on the scale is .0001". Fig. 740 shows a common type of electric amplifier and the standards by which it is set. To use the amplifier, Jo-blocks are stacked to the size of the dimension to be checked. The blocks are then placed on the amplifier block and the amplifier adjusted so that when the gaging point contacts the blocks, the indicator pointer will have a deflection of not more than 10 divisions and so the pointer rests on zero. The Jo-blocks are then removed and the work placed under the contact point with a sliding or rolling motion. Any variation between the Jo-block size and the work size is then visually recorded in tenths of thousandths on the dial or scale.

TAPER WORK

ACCURATE TAPER WORK may be produced on the cylindrical grinders by one of two methods, either by swiveling the work table or the headstock. For slight tapers the

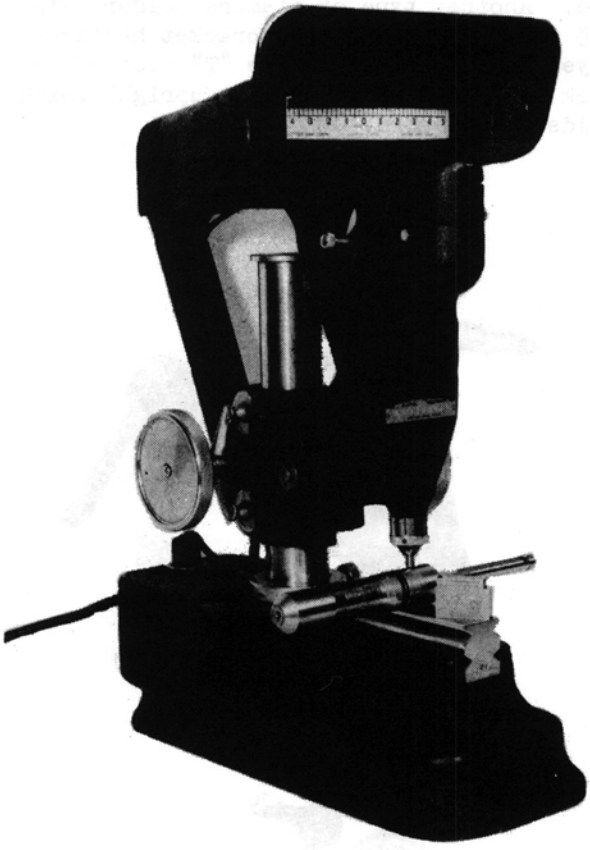


Fig. 740

table may be set by means of the swivel adjustments and table graduations, but inasmuch as this can only be approximately accurate, a standard taper ring gage, female taper gage, or sine bar is necessary to check the taper. The grinding of tapers is very much the same as grinding a cylinder, the exception being that the swivel table or wheel stand slide is set to produce the correct taper angle. Generally the graduations on the scale marked degrees, is one-half of the whole taper angle, while taper per foot or percentage indicates the whole taper angle.

If a taper ring gage is used for checking the accuracy of the taper, Fig. 741,

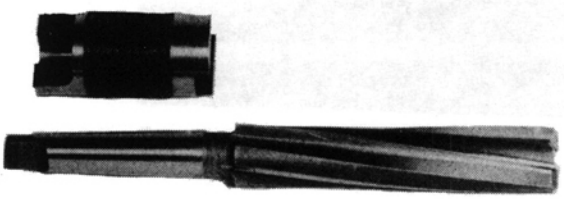


Fig. 741

the male section should be given three lengthwise stripes of prussian blue about 120° apart and then carefully inserted into the ring gage with a slight twisting motion. If the surface being checked does not conform to the surface of the gage, the irregularity rubs the blue off and leaves a bright metallic ring indicating the high spot. If the gage bears only on one or two lines it indicates that the piece being tested is out of round.

STEEP TAPERS can be produced by swiveling the work head or by dressing the wheel at an angle. If the wheel is to be dressed at an angle it is absolutely necessary to have the diamond set on the exact centerline of the wheel. This is necessary, not only to insure dressing the correct angle on the wheel, but also to obtain a straight face on the wheel rather than one which is concave or convex.

The 60° point on a machine center is ground (Fig. 742) by placing it in the live

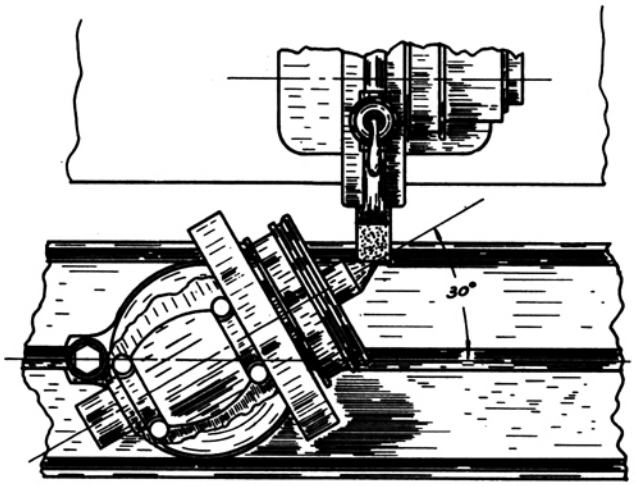


Fig. 742

spindle of the headstock and swiveling it through a 30° angle using the graduations on the base of the headstock. It is obvious that these graduations cannot be relied upon for extreme accuracy, therefore by using the flat center gage, Fig. 743, or the bell center gage, Fig. 744, as a check, the necessary adjustments can be made to insure accurate results.

The actual operation of grinding machine centers must be carried out with care as there is danger of burning the point because of the change in work speed due to the steep taper. This trouble may be overcome by using a slow work speed and small infeed, a flood of coolant, and starting the cut

at the point, ripping it back towards the shank.

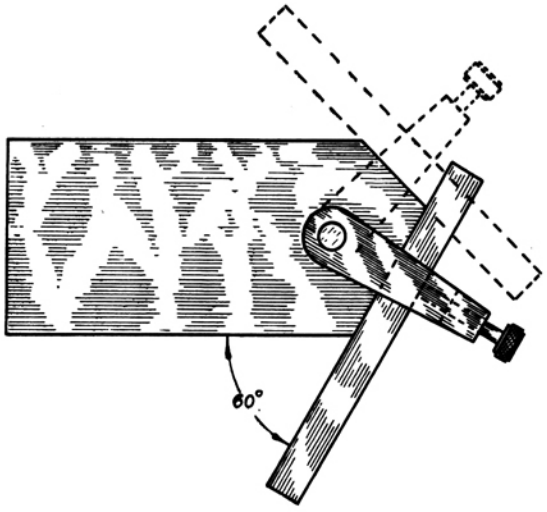


Fig. 743

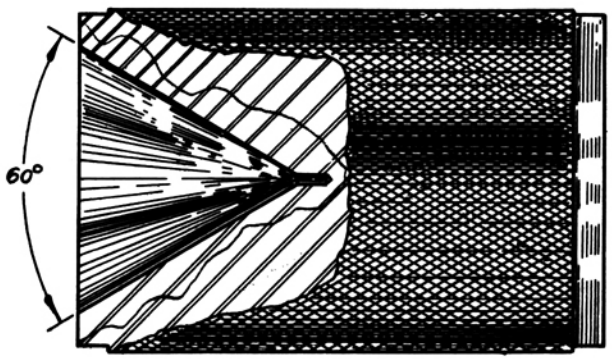


Fig. 744

WHEEL DRESSERS

On all external grinders some provision is made for dressing the wheel. The footstock is equipped with a suitable holder, so arranged that it is adjustable and can be located on the centerline of the wheel spindle. Another type of diamond holder (Figs. 745 and 746) is a forged bracket having a keyed base to fit into the "T" slot of the work table and an adjustable upright which holds the diamond.

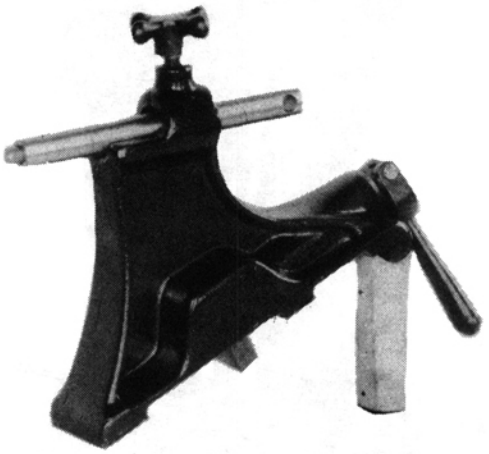


Fig. 745

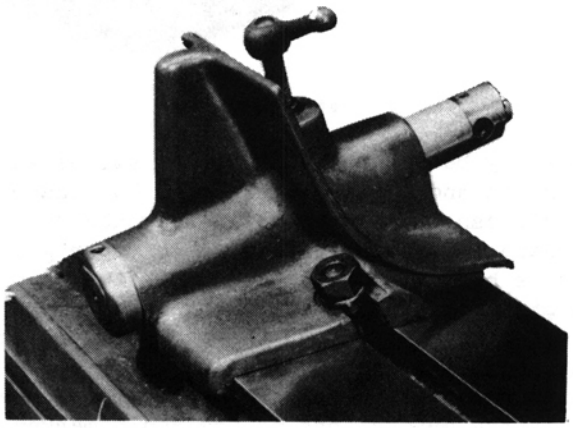


Fig. 746

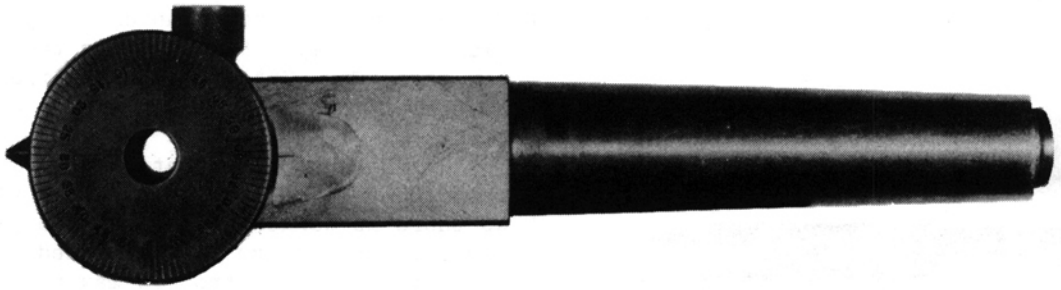


Fig. 747

Another type known as a micrometer adjustment wheel dresser (Fig. 747) consists of a hollow screw body surmounted by a dial graduated to read in thousandths of an inch, which passes through a threaded hole located on the axis of the footstock center. The diamond is inserted in the hollow screw and locked in place by a set screw. This dresser proves to be very efficient especially where the job consists of a number of pieces of the same size. To use it, after the first piece has been reduced to size, the point of the diamond is spotted on the grinding wheel so that the distance from the axis of the footstock center to the tip of the diamond is equal to the radius of the work. The micrometer screw is then locked in place insuring the correct sizing ability of the wheel.

SAFETY ITEMS

It is important when operating an external grinder to let the machine run for a few minutes when first starting it, to give it a chance to warm up. During this time let the coolant run on the wheel to balance it, because having stood over night the coolant drains to the bottom of the wheel causing it to be out of balance. Be sure to check the stops, feed trips, and levers to make certain that the wheel doesn't run into the machine and damage it. See that the driving pin between the drive plate and grinder dog is securely locked in place and proper adjusted. Keep a good stream of coolant running at the point of contact between the wheel and work as it helps to dissipate the heat, tends to give the work a better finish, and keeps the wheel clean. Keep your hands away from the moving wheel and work. If something goes wrong stop the machine by pulling the main switch and call an instructor.

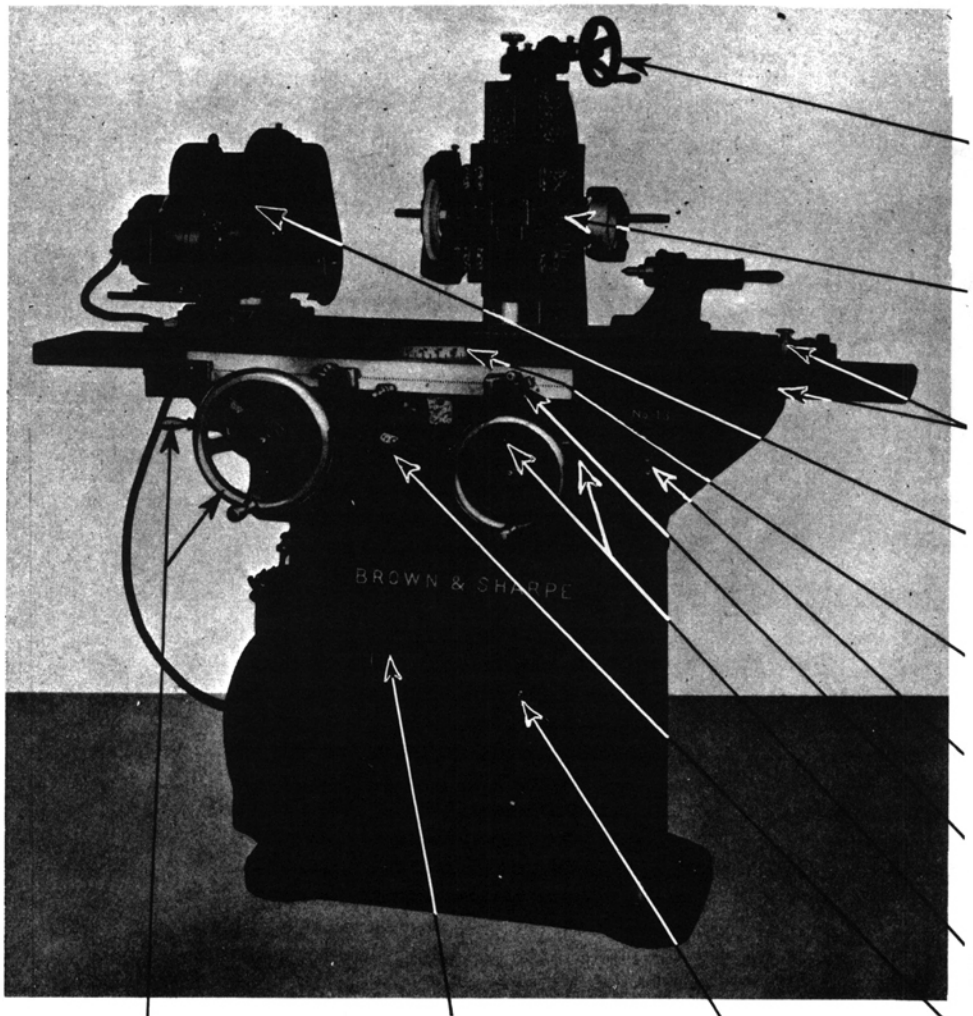

BROWN & SHARPE
Elevating screw handwheel swivels to any convenient operating position. Bracket clamped in position by knob. Handwheel graduated to read to .0005"
Double-ended wheel spindle carried in sturdy slide. Has vertical adjustment of 8". Proper adjustment of spindle boxes automatically maintained by spring shoes.
Spring latch, and knob for fine adjustments of swivel table. Latch engages knob.
Motor-driven headstock has both dead-center and revolving-spindle drive. Swivels on graduated base. Lever operates quick-acting clutch.
Swivel table turns on stud to 90° either side of zero. Scale, graduated to degrees, indicates setting.
Main start-stop push button conveniently located. Controls both motors.
Table reversing dogs quickly set. Have fine thumbscrew adjustment.
Fine cross feed operated by small handwheel; engaged by knob on cross feed handwheel.
Table reversing lever. Can be operated by hand. Lock prevents engagement of longitudinal power travel when handwheel is in use.
Lever for selecting fast hand travel, slow hand travel, or automatic power travel of table; and longitudinal table handwheel.
Separate start-stop switch and overload relay reset for headstock motor. All electrical controls enclosed in base.
Reset for wheel spindle and table motor overload relay.

Fig. 748. Universal and Tool Grinding Machine

UNIVERSAL GRINDERS

Besides the external grinder just discussed another grinder which is rapidly growing as a utility, is the Universal Tool Grinder (Fig. 748). This machine is one which can, within its capacity, do the work of other grinding machines provided the necessary attachments are available. Because of its adaptability, some shops use nothing but universal grinders, other shops use them for work which might interfere with continuous operation of specialized machines. This machine is truly universal; the headstock is adjustable both horizontally and vertically, the wheel head can be swiveled in a horizontal plane and can be raised or lowered, and the table can be traversed and swiveled.

CENTERLESS GRINDERS

The Centerless grinder (Fig. 749) is a specialized machine which was developed for the rapid production of cylindrical, external

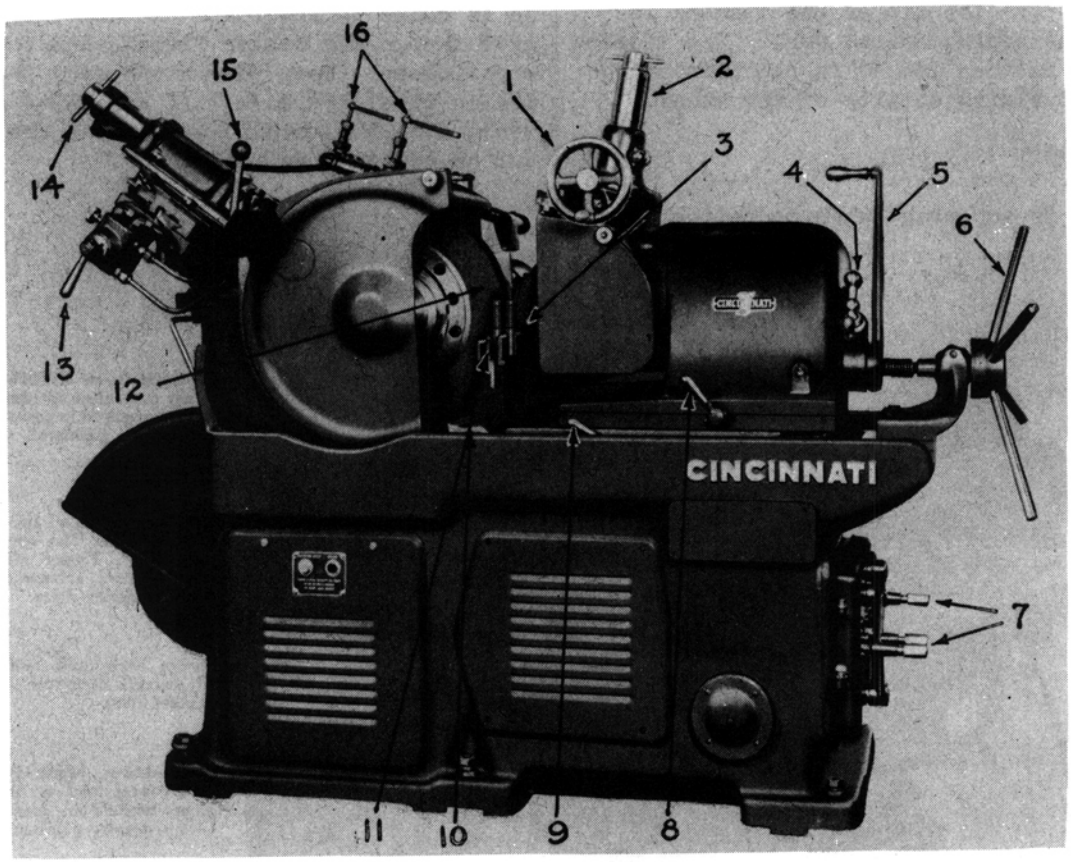


Fig. 749. Centerless Grinder

1. Handwheel for operating regulating wheel truing attachment.
2. Regulating wheel truing attachment.
3. Regulating wheel.
4. Ball crank for micrometer adjustment of regulating wheel.
5. Hand lever for adjusting pressure on regulating wheel.
6. Spar for adjusting regulating wheel and work rest slides.
7. Speed change levers controlling regulating wheel speed.
8. Lever for clamping regulating wheel slide to lower work-rest slide.
9. Lever for clamping lower work-rest slide in position.
10. Upper work-rest slide for transverse adjustment.
11. Work-rest.
12. Grinding wheel.
13. Lever for hydraulic control of grinding wheel truing attachment.
14. Grinding wheel truing attachment.
15. Lever for operation of profile attachment.
16. Valve handles for controlling coolant supply to diamond and grinding wheel.

tapers, or external profile work (Fig. 750). In centerless grinding two wheels are employed; one, the cutting or grinding wheel, is used to remove the excess stock; the other, a "regulating wheel," which is used to control the speed of rotation of the work and rate of feed. The work is supported on a work slide or rest.

This machine has many distinct advantages over other grinders in that the work does not have to be center drilled, thereby saving the time required for that operation on the lathe; since the work does not have to be mounted on centers and since the grinding operation is almost continuous, the time required for loading and unloading is saved; heavier cuts can be taken and the amount of material to be removed can be less, therefore saving more time and adding to the life of the grinding wheel; the operation of the machine does not require the services of a skilled mechanic and since there are few moving parts in the machine, the upkeep cost is very low, and the output rate very high.

It is generally thought that because the external centerless grinder was designed for jobs of a large number of pieces, that it is not suited for tool room use, but actually much time and money can be saved where the job lot contains only a few pieces because the machine is simple to set up.

The actual grinding operation depends for the most part on the pressure exerted by the grinding wheel on the work and the operation of the work with respect to the wheel centers (Fig. 751). In operation, the pressure exerted by the grinding wheel on the work forces the work against the work rest and regulating wheel. The regulating wheel revolves in a direction the same as that of the grinding wheel and has a horizontal movement. It has speed of 12 to 300 R.P.M., and at the same time feeds the work through the machine. The rounding of the work depends on how high the work rests are above the

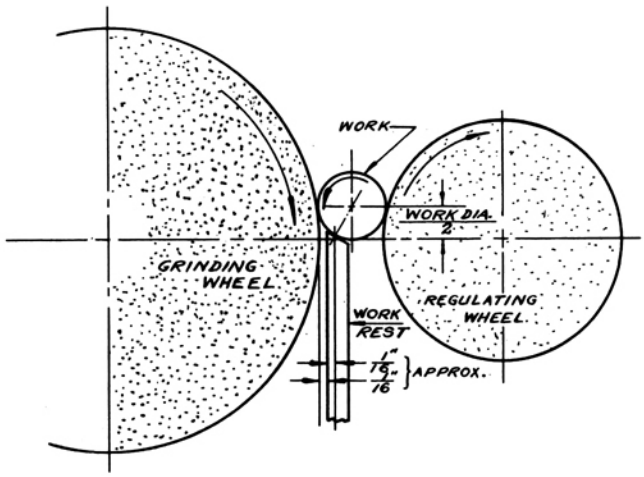


Fig. 751

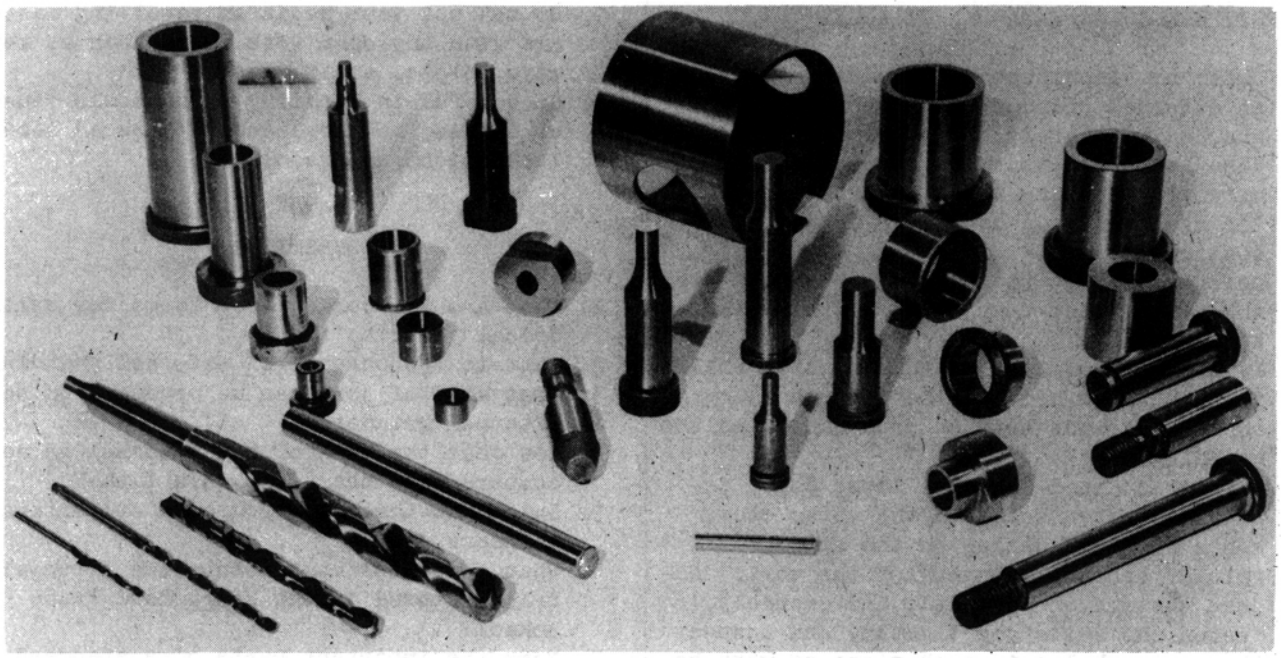


Fig. 750

centerlines of the wheels and the top angle of the work rest.

METHODS OF FEEDING

Two common methods used for feeding the work to the machine are the THROUGH FEED and IN FEED. The through feed method is used on straight cylindrical work whereas the in feed method is used for shoulder, taper, or profile work. See Figs. 752 and 753.

"The center of the work should be located approximately one-half the work diameter above the centerline of the wheels."

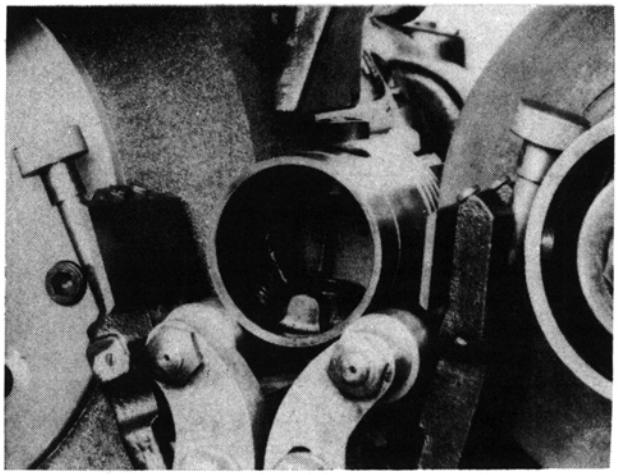


Fig. 752. Through Feed

SAFETY RULES FOR EXTERNAL GRINDERS

1. Under no circumstances attempt to operate an external grinder unless the wheel is guarded adequately.
2. When it is necessary to replace a large external grinding wheel, ask the instructor about methods of proper mounting and testing.
3. Before starting the work head, always test the work to see that it is between centers.
4. If the work must be tested for size while it is between centers, be sure to allow ample clearance between your hands and the grinding wheel.
5. Poorly adjusted drivers, loose dogs, defective center holes in the work, etc., are a constant danger on the external grinder causing spinning of the work. Adjust the driver correctly and securely, fasten the drive dog tightly, and inspect the center holes in each piece of work before grinding it.
6. Be extremely careful in removing work from a collet head. Run the table back to a SAFETY STOP which will give ample hand clearance between the wheel and work. (ASK THE INSTRUCTOR ABOUT THIS).
7. If the work is heavy, shut the machine down when placing it between centers.
8. Avoid dressing the sides of a large grinding wheel, if possible, but if it is necessary for the job, ask the instructor to help. He will demonstrate the right method of dressing the wheel and how cuts can be taken so as to preserve the corners of the wheel.

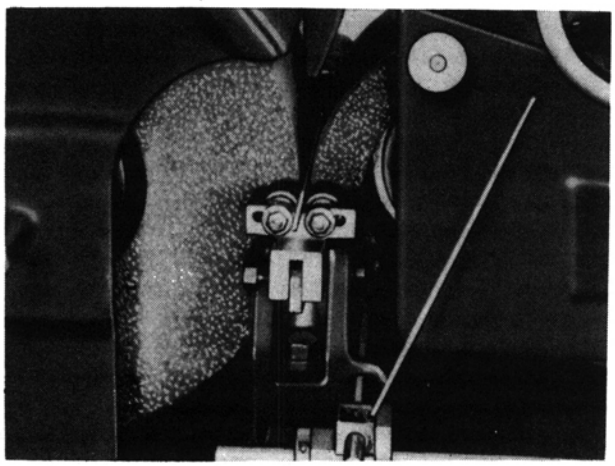


Fig. 753. In Feed

9. Do not put your hands on revolving materials with any open work in it such as keyways, slots, and flutes.
10. Be careful in handling sharp tools, such as drills, reamers, and cutters as severe lacerations may result.

QUESTIONS

1. What are the common shop terms for cylindrical grinding?
2. What is the purpose of external grinding?
3. What kind of jobs can be produced on the external grinder?
4. How does the work of the external grinder compare with that done on a lathe?
5. Into what three major groups are external grinders divided?
6. What companies manufacture the external grinders used in the Henry Ford Trade School?
7. Name the important parts of a standard external grinder.

8. What job operations can be done on the plain cylindrical grinder?
9. What movements are important in any cylindrical grinder?
10. Why are each of these movements important?
11. Explain how the job in external grinding is driven when supported between centers.
12. Generally speaking what should the length of the table traverse be?
13. Why is it important to permit the grinding wheel to over-run the end of the job?
14. What happens to the job if the grinding wheel completely over-runs the end of the work?
15. What does the speed of the table traverse depend upon?
16. How fast should the speed of the table traverse be?
17. How is long, slender work supported while being driven between centers?
18. What care should be exercised in the use of steady rests?
19. What provisions are made for feeding the grinding wheel of the work?
20. What depths of cuts is it possible to take with the automatic feed?
21. Why is it advisable to use the automatic feed rather than the hand feed?
22. What is the depth of a roughing cut using the automatic feed?
23. What speed and feed should be used for roughing out a job? For finish grinding a job?
24. Give the grades of bond, size, and kind of abrasive commonly used for external grinding wheels?
25. Why is it not possible to have a set rule for the rate of work speed?
26. What work speeds are commonly used?
27. What conditions might cause cracks, checks or burns on the surface of the job?
28. What conditions might cause tapering of the work?
29. How can tapering of the work be overcome?
30. What is meant by "plunge cut" grinding?
31. When, in particular, should plunge cut grinding be used?
32. Why should the job be checked for size before setting the job up?
33. Footstock centers which are cut away are called what kinds of centers?
34. What care must be exercised in attaching the grinder dog to the end of the job?
35. How should the diamond be used to make the wheel fast cutting?
36. When the wheel is first fed to the work what notice should be taken of the work contact?
37. Why should the work be checked after the first cut is taken?
38. How should the diamond be used to prepare the wheel for the finishing cut?
39. If the work has keyways or splines, how should it be prepared if steady rests are to be used?
40. How is it possible to grind a cylinder completely at one time without using a grinder dog?
41. If the work has several shoulders, what is the proper way to grind it? Why?
42. Why can micrometers not be used for gaging jobs requiring a high degree of accuracy?
43. What is an "amplifying comparator"?
44. Explain how to use the amplifier.
45. To what degree of accuracy can the ordinary amplifier be used?
46. How is it possible to produce taper work on the cylindrical grinder?
47. What tools are necessary for producing accurate taper work?
48. How is the taper of the job generally specified?
49. Explain how to use a taper ring gage for producing an accurate taper.
50. In producing steep tapers what care must be exercised in dressing the wheel?
51. Explain an efficient method of grinding machine centers.
52. How may the accuracy of the 60° conical point of the machine center be checked?
53. Name three kinds of diamond holders used on external grinders.
54. What advantage does the micrometer adjustment diamond holder have over the others?
55. Give six important safety rules for operating external cylindrical grinders.
56. What is meant by a "Universal Tool Grinder"?
57. What is the advantage of this machine?
58. What are its disadvantages?
59. What kinds of work can be done on the Centerless grinder?
60. What is the principle of operation of the centerless external grinder?
61. What advantages does the centerless grinder have over the plain and universal external grinders?
62. Upon what two factors does the actual grinding operation depend in centerless grinding?
63. In what direction does the regulating wheel rotate with respect to the grinding wheel?
64. Upon what two factors does the rapidity of the rounding of the work depend?
65. What is meant by "through feed"? "In feed"?
66. What is meant by "profile work"?

SURFACE GRINDERS

Surface grinding is the act of producing and finishing flat surfaces by means of a grinding machine employing a revolving abrasive wheel.

Surface grinding machines are divided into two major groups according to the shape of the table and how it moves. They are, the "planer type" where the table is rectangular in shape and traverses under the wheel, and the "rotary type," where the table is circular in shape and rotates under the wheel.

PLANER TYPE	EXAMPLE
1. Horizontal spindle using the O.D. or periphery of the wheel.	(#3B Abrasive)
2. Horizontal spindle using the rim of a recessed, or cupped wheel.	(Gardner Surface)
3. Vertical spindle using the rim of a recessed, or cupped wheel.	(6 X 18 Norton Surface)

ROTARY TYPE	EXAMPLE
1. Horizontal spindle using the O.D. or periphery of the wheel.	(Heald Rotary #22)
2. Vertical spindle using the rim of a recessed, or cupped wheel.	(#18 Blanchard)

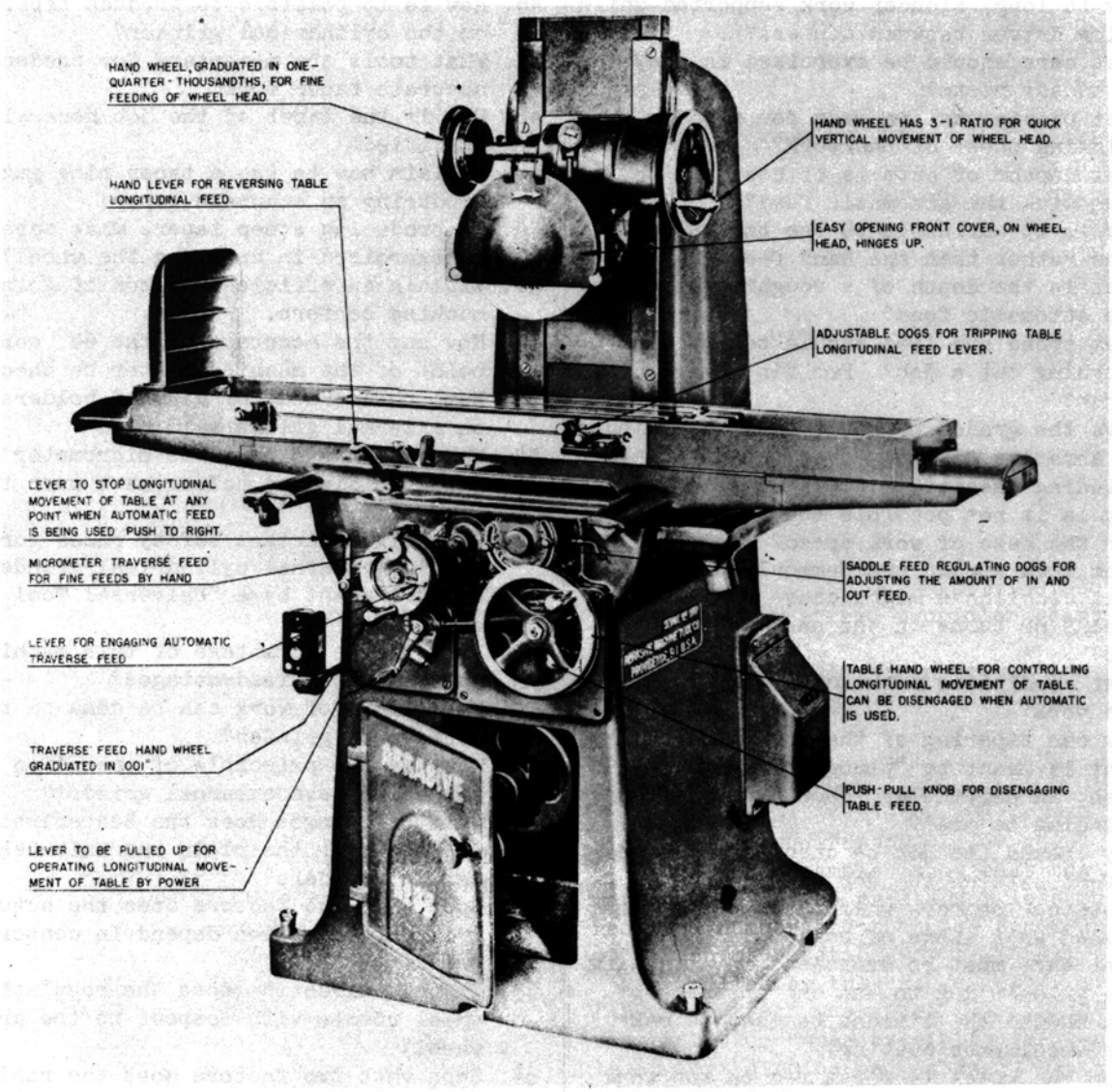
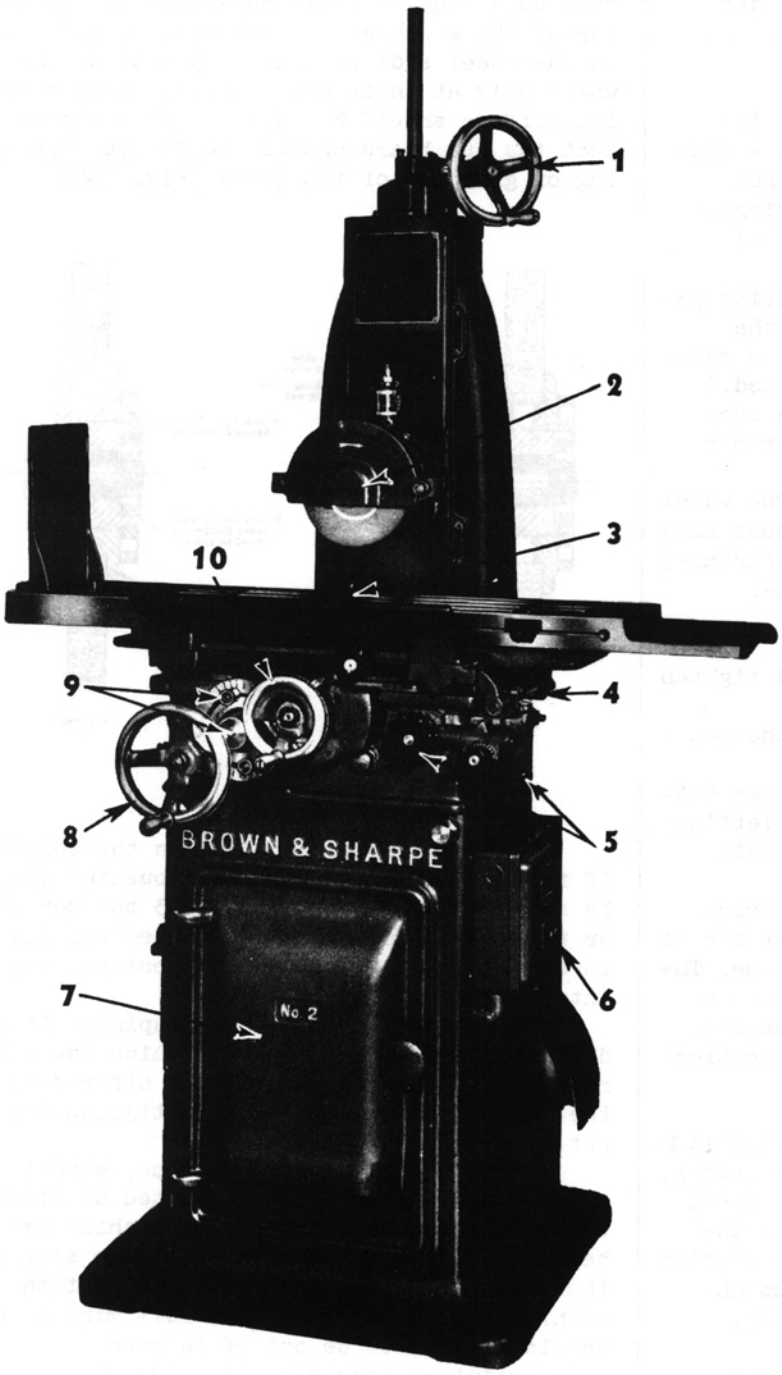

HAND WHEEL, GRADUATED IN ONE-QUARTER-THOUSANDTHS, FOR FINE FEEDING OF WHEEL HEAD.
HAND WHEEL HAS 3-1 RATIO FOR QUICK VERTICAL MOVEMENT OF WHEEL HEAD.
HAND LEVER FOR REVERSING TABLE LONGITUDINAL FEED.
EASY OPENING FRONT COVER, ON WHEEL HEAD, HINGES UP.
ADJUSTABLE DOGS FOR TRIPPING TABLE LONGITUDINAL FEED LEVER.
LEVER TO STOP LONGITUDINAL MOVEMENT OF TABLE AT ANY POINT WHEN AUTOMATIC FEED IS BEING USED. PUSH TO RIGHT.
MICROMETER TRAVERSE FEED FOR FINE FEEDS BY HAND.
SADDLE FEED REGULATING DOGS FOR ADJUSTING THE AMOUNT OF IN AND OUT FEED.
LEVER FOR ENGAGING AUTOMATIC TRAVERSE FEED.
TABLE HAND WHEEL FOR CONTROLLING LONGITUDINAL MOVEMENT OF TABLE. CAN BE DISENGAGED WHEN AUTOMATIC IS USED.
TRAVERSE FEED HAND WHEEL GRADUATED IN .001".
PUSH-PULL KNOB FOR DISENGAGING TABLE FEED.
LEVER TO BE PULLED UP FOR OPERATING LONGITUDINAL MOVEMENT OF TABLE BY POWER.
ABRASIVE

Fig. 754. Planer Type Grinder

Fundamentally any of the above mentioned machines consists of a spindle with a grinding wheel mounted on it, and a table or magnetic chuck for holding the work for the purpose of presenting it to the wheel. Each machine has its particular advantages but for the purpose of general tool room work this discussion deals principally with the horizontal planer type of surface grinder (Figs. 754 and 755) which uses the periphery of a disk wheel.

SURFACE GRINDER WHEELS

Grinding wheels of various shapes are used in the horizontal surface grinder, among the more common being the disk, gage, and thin rubber slotting wheels. The sizes of the wheels may vary from very small wheels of the internal class, which may be used with a high speed attachment, to wheels 10" in diameter having a 1" face. The wheels most commonly used on surface grinders are:



1 Handwheel, graduated to half-thousandths of an inch, provides easy and accurate vertical adjustment of spindle head.

2 Wheel spindle runs in phosphor bronze front box and ball bearing rear support. Front box has means of compensation for wear. Heavy wheel guard with removable cover gives ample protection.

3 Table reversing mechanism operated manually through lever shown, or automatically through second lever located on same shaft and tripped by adjustable table dogs.

4 Adjustable dogs permit stopping power movements automatically at any desired point in each direction of cross feed.

5 Knob and trip lever start and stop both longitudinal and transverse power movements. Turning knob to right starts power feed; depressing lever stops feed instantly.

6 Start-stop push button and overload relay reset conveniently located.

7 Large base compartment has shelf for mounting driving motor. On overhead-driven machine, provides handy storage space for Attachments and accessories.

8 Longitudinal table handwheel conveniently located and easily operated. Can be positively disengaged when power travel is used.

9 Adjustable stops provide for any cross feed from .01" to .09" (or zero feed) at either end of table travel. Knob in center disengages cross feed mechanism for manual operation.

10 Graduated handwheel permits rapid and accurate transverse adjustment.

Fig. 755. Planer Type Grinder

46-G; 46-H; 46-I; 46-J; 46-K.

60-H; 60-I; 60-J; 60-K.

Such materials as stone, glass, copper, or tungsten carbide require grinding wheels of special grain, material, and structure.

MOUNTING SURFACE GRINDER WHEELS

The following items are to be considered in mounting a grinding wheel on the horizontal planer type of surface grinder from the time the wheel is received at the crib until the machine is running:

1. Sound the wheel for cracks. Hold the wheel by the bore and tap it with a non-metallic object; if the wheel is not cracked it will give off a dull ringing sound; if the wheel is cracked it will give a dull thud.
2. Make sure that the wheel has blotting paper gaskets on both sides around the hole. (Most wheels arrive from the manufacturer with paper washers attached.)
3. Inspect the wheel flanges to make sure they are of the safety type and proper size.
4. Place the wheel on the spindle. The wheel should slide on the spindle, without bind or too much play (.003 to .005 clearance) up against the inner safety flange.
5. Put the outer safety flange on.
6. Put the spindle nut on securely.
7. Put the wheel guards in place and tighten them.
8. Make sure that the wheel clears the housing by turning it over by hand.
9. Start the machine and as the starter button is pressed step to one side, letting the machine run for at least one full minute before working with it.
10. Truing the wheel. Don't try to remove all of the runout with one pass of the diamond. Remove it a little at a time. The wheel speed has been built into the machine by the manufacturer. For safety, don't try to increase it without getting competent advice.

From a safety standpoint of view it is very essential to sound the wheel for cracks, because they may not be seen. If the wheel is cracked, and placed on the machine, the centrifugal force when the machine is started up will cause the wheel to burst, thus endangering the operator or near-by fellow workers.

Blotting paper or rubber gaskets should be placed between the safety flanges and the wheel to evenly distribute the pressures around the wheel when the nut is tightened.

80-O 120-P

$\frac{1}{32}$ to $\frac{3}{32}$ rubber slitting wheels.

SAFETY FLANGES

Safety flanges are wheel flanges which act on the wheel between the spindle and nut. They should be at least one-third the diameter of the wheel and be relieved or undercut on the wheel side so that they bear on the wheel only at their outer edges. This bearing surface should be parallel with the side that the nut bears against to insure even holding or gripping of the wheel (Fig. 756).

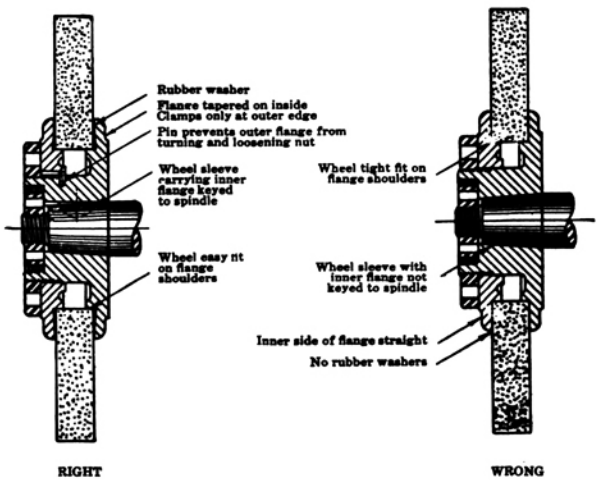

Rubber washer
Flange tapered on inside
Clamps only at outer edge
Pin prevents outer flange from turning and loosening nut
Wheel sleeve carrying inner flange keyed to spindle
Wheel easy fit on flange shoulders
Wheel tight fit on flange shoulders
Wheel sleeve with inner flange not keyed to spindle
Inner side of flange straight
No rubber washers
RIGHT
WRONG

Fig. 756

Never force the wheel on the spindle. If it goes on snugly, the lead bushing should be scraped so as to be from .003 to .005 larger than the spindle, which enables the flanges to straighten the wheel up without putting an internal strain on it.

The nut threads on the spindle in a direction opposite to that in which the wheel rotates, so that the resistance offered by the work to the wheel tends to tighten the nut.

When starting the machine, especially after a new wheel has been mounted or first thing in the morning after the machine has been standing all night, step to one side as the starting button is pressed and let the machine run for at least one full minute. New wheels are apt to be out of balance due to being moisture logged on one side or the wheel spindle may have excessive end play in

it which upon starting would break the wheel. For that reason be clear of it in case it does break.

DRESSING SURFACE GRINDER WHEELS

In the case of surface grinding where the work is ground dry it is very necessary to keep the wheel sharp and true if the grinding operation is to be done efficiently. The area of contact between the wheel and work is much greater than it is in cylindrical grinding, which means that a great many more cutting tools are acting on the work at the same time. This means that the heat generated due to the added cutting tools must be kept at a minimum by keeping the wheel clean and sharp.

A dull or dirty wheel causes the work to burn or have hard cutting action. Hard cutting action means more strain on the wheel and machine parts causing the wheel and machine parts to heat up and operate inefficiently.

To dress a wheel it is merely necessary to remove sufficient material from the face of the wheel to sharpen it. Don't keep feeding the wheel down and hack away at it with the diamond. Grinding wheels are costly tools and should be shown due consideration. Some grinding wheel manufacturing companies claim that about two-thirds of a grinding wheel is wasted due to improper dressing, truing, or handling, so use it accordingly.

HOLDING WORK

Most surface grinders employ a magnetic chuck (Fig. 757) for holding the work, but it may be held by clamping it directly to the table or by clamping it in a vise and

Fig. 757

bolting it to the table. Work may be held in conjunction with the magnetic chuck by using a "V" block, angle plate, sine bar, or special fixtures.

The magnetic chuck holds the work by exerting a magnetic force on it. The magnetic poles of the chuck are placed close together so that it is possible to hold very small pieces of work. Frequently, however, the work is too small for the chuck to hold it, so that long pieces of iron or steel, having a height less than that of the work, are placed along side to act as blocks thus retaining the work on the chuck. It must be remembered that only magnetic materials such as iron or steel will actually adhere to the chuck. When grinding such materials as bronze, brass, fibre, or certain kinds of stainless steel, the pieces must be held in place on the chuck by bars of iron or steel.

The accuracy of the work, such as parallelism, squareness, etc., depends on the accuracy of the holding face of the magnetic chuck. For that reason the holding surface of the chuck must be kept smooth and flat. As soon as the chuck shows nicks, scratches, or dents, or if the chuck has been removed for any purpose, it should be reground, in place, with care. Do this only on the advice of the instructor or foreman.

Generally the chuck is equipped with a back rail, the purpose of which is to support the work parallel to the table travel. This is removed when the chuck is reground, so that when it is replaced it must also be ground in place to restore its accuracy.

Few jobs are ever done on a surface grinder in the tool room, which do not have to be kept parallel within reasonably close limits. The most frequent cause of difficulty in obtaining parallelism is a dirty or poorly kept work surface on the chuck or the back rail not being parallel with the table travel. On thick work, if difficulty is experienced in keeping the opposite faces parallel, it may be overcome by reversing the position of the work on the chuck, putting it in nearly the same location, but without disturbing the wheel setting.

Thin work is especially hard to keep parallel because it warps so easily. The use of a free cutting wheel and light cuts taken alternately from each side do much to eliminate the warping and hence make it easy to keep the work parallel within reasonable limits. Another method is to place a thin parallel under each end of the work, taking a light cut or series of light cuts alternately from each side. When using this method, the work should be properly blocked on the table to keep it from sliding.

WITNESS MARKS

Occasionally the surface of a job is required to be ground "to clean up." By this is meant that the surface is to be ground so that a very small portion of the original

surface can be seen. This is known as leaving "witness marks" on the work.

The purpose of leaving these witness marks is to testify that only a small amount of stock was removed from the job. In some jobs this means that the wheel must be spotted on the work and then traversed over the entire surface to make sure that too much stock is not removed. If the surface is quite regular and has a good finish the edges may be given a coat of blue vitriol or copper sulphate to show that too much stock was not removed.

ANGLE PLATE WORK

90 degree angle plates (Fig. 758) are used on the surface grinder for supporting the work while two adjacent faces are ground at 90° to each other.

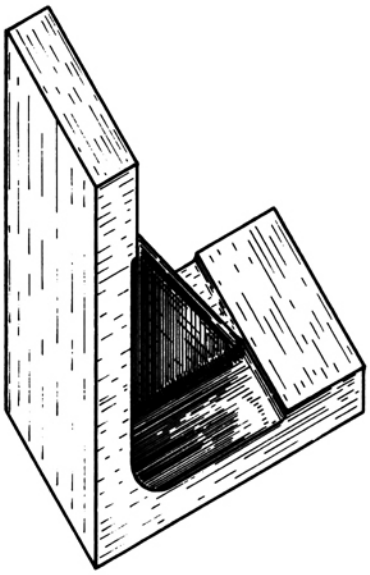


Fig. 758

Obviously the accuracy of the work will depend on the accuracy of the plate. If the two adjacent faces of the plate are ground at exactly 90° and if the edges of the plate are maintained parallel and at 90° to the adjacent faces, then it is possible to grind two sides of the work at 90° with a third side, thus cutting down on the work set-up time.

ADJUSTABLE ANGLE PLATES (Fig. 759), precision made, can be had which make it possible to grind intricate angles in the work. Some of these plates, known as "Sine Angle Plates," and which have provisions for setting them with a micrometer, or Johansson gage blocks, are quite suitable for grinding angles which have exacting limits. The "Sine bar" is frequently used for grinding angles where an adjustable angle plate is not available.

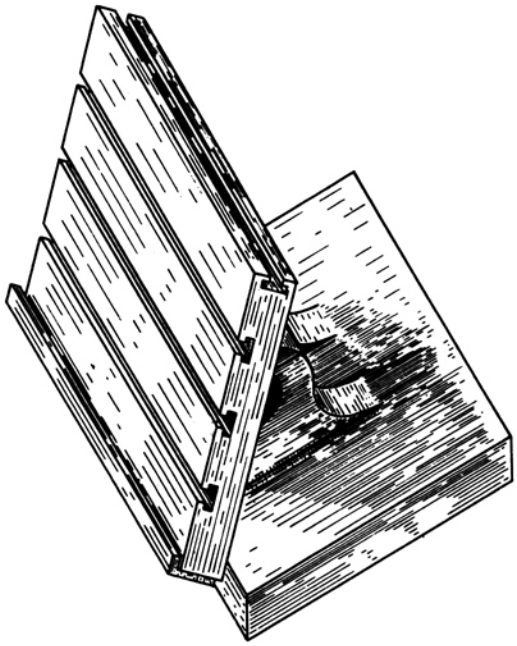


Fig. 759

Fig. 760

GRINDING WORK SQUARE

To grind the work square after two

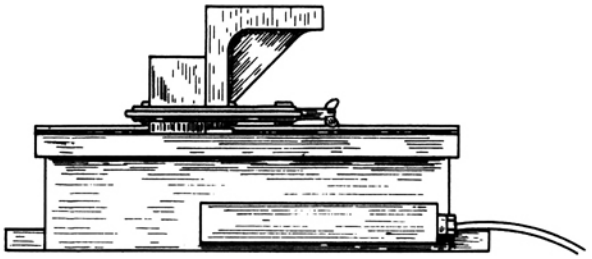


Fig. 761

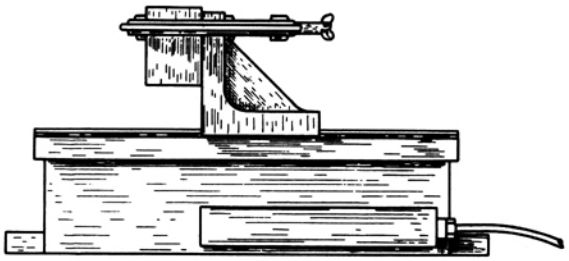


Fig. 762

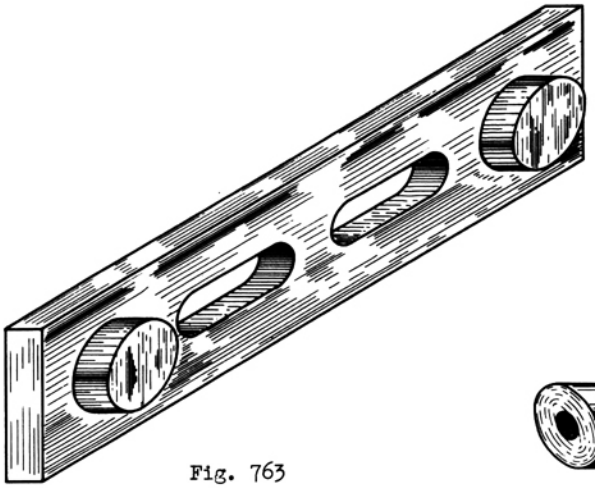


Fig. 763

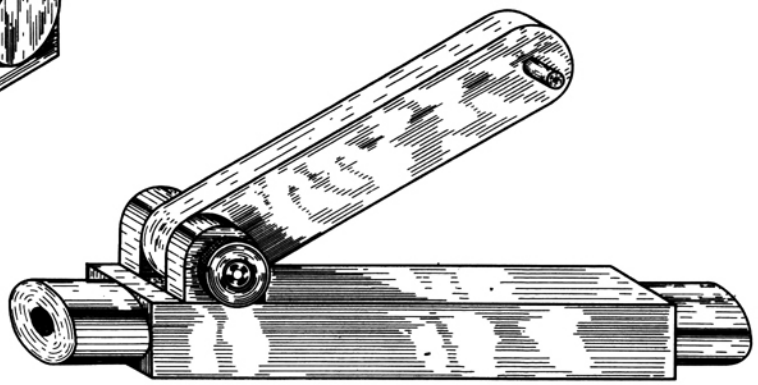


Fig. 764

sides have been ground parallel, place the side to be ground, down on the magnetic chuck. Place a thin parallel or rule alongside of the work, then place the 90° angle plate, top edge down on the rule, bringing the outside face of the plate up against the finished side of the work (Fig. 761). Clamp the work and plate together in this position, so that when the angle plate is turned right side up on the magnetic chuck (Fig. 762) the work face to be ground will be exposed to the grinding wheel. Make sure that the clamp has been placed far enough below the face to be ground and that the screw end is located inside the angle plate to reduce the overhang and so as not to interfere with the grinding operation.

ANGULAR SURFACES can be readily ground on a surface grinder by dressing the wheel to produce the required angle, by using a sine bar (Figs. 763, 764, and 765) or adjustable angle plate and setting the work up to the required angle or by using an adjustable magnetic chuck.

The grinding wheel should not be dressed at an angle except as a last resort, because it decreases the wheel life. If the wheel must be dressed at an angle make sure it is dressed correctly, that is, dress the wheel so that it will produce the angle required on the work and not the complement of it. For dressing the wheel at an angle, either a standard angle block or sine bar may

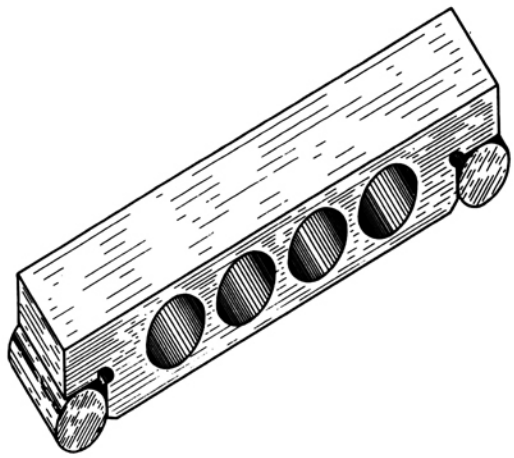


Fig. 765

be used to guide a sliding flanged block which holds a diamond as shown in (Fig. 766). After the angle block or sine bar has been set up it is located so that the point of the diamond is exactly on the vertical centerline of the wheel spindle. See (Fig. 767).

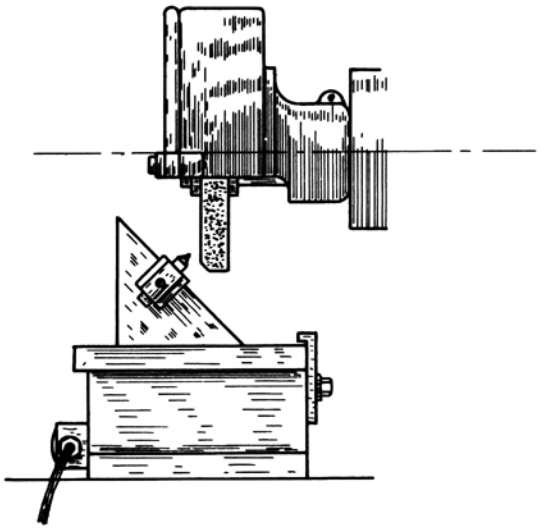


Fig. 766

When the work is to be set on an angle the common practice is to use a standard angle block or sine bar (Fig. 768) in conjunction with a right angle plate. With some sine bars all that is necessary is to stack gage blocks or set a planer gage at 5 or 10 times the sine of the angle, depending on the length of the sine bar; then place the work on the sine bar squarely against the 90° angle plate and clamp it in place. Other sine bars require the addition of the radius of plugs or thickness of the base of the sine bar be added to 5 or 10 times the sine of the angle.

V-blocks are used for holding cylindrical work while grinding flats, slots, radii, etc. The blocks used for this purpose are kept square, the sides parallel, the ends parallel, and

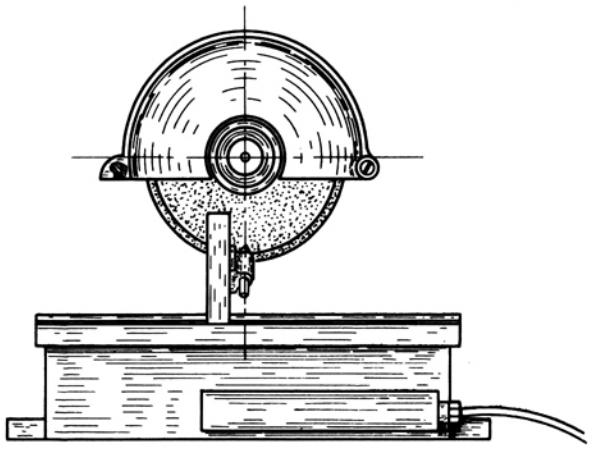


Fig. 767

the "V" maintained in the exact center of the block.

JOB ANALYSIS

The following outline is a typical analysis of the procedure for grinding two opposite flats central on the OD of a shaft.

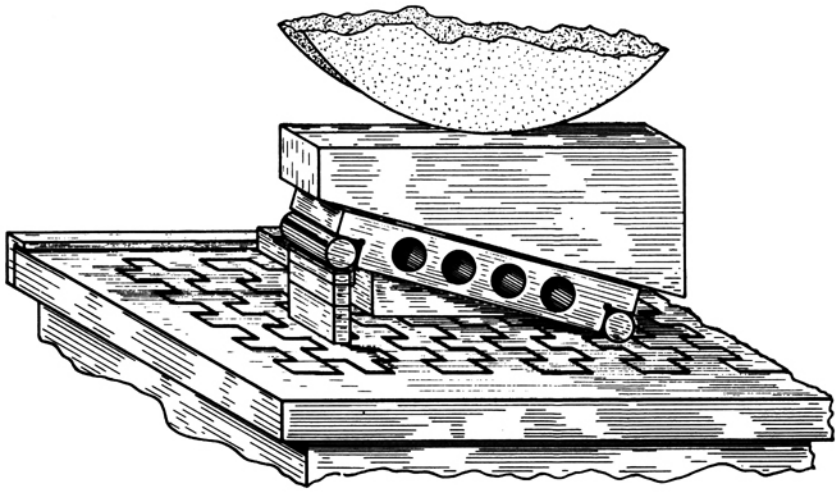
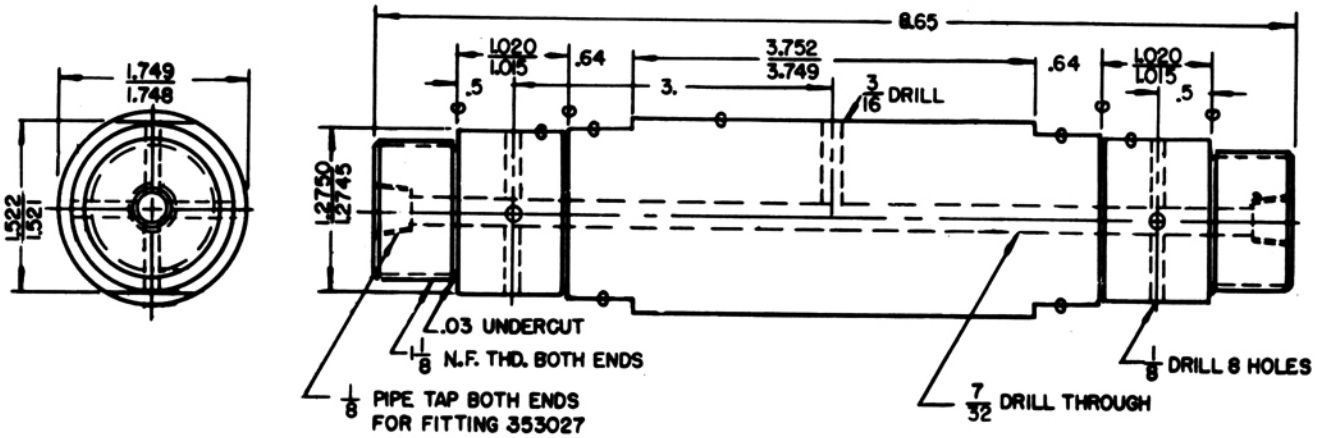


Fig. 768

Type of job. . . . . shaft as per sketch, (Fig. 769(a)).
Type of machine . . . . . Horizontal planer type.
Type of steel. . . . . ."RR".
Heat treatment . . . . . "Q".
Kind of wheel . . . . . 60 - K.
Operations required . . . . . Grind flats to size and central with OD.
Tools required . . . 4" V-block, 10" height gage, 1-2" mics, Ideal indicator.



DET-56 SHAFT
630 MACHINE STEEL
STK. 2. DIA. WT. #7 LBS.
HEAT TREATMENT "P"
ROCKWELL 64 65
GRIND

Fig. 769(a)

PROCEDURE

1. Check flats for sufficient grind stock.
2. Place the job in the V-block using a 10" height gage and Ideal indicator so that the rough flats of the job are parallel with the sides of the block.
3. Using the height gage and indicator, check the cylindrical surface of the job projecting from the V-block for being central. This is accomplished by indicating the sides of the job as shown in Fig. 769(b) turning the V-block from side to side and observing if the reading is the same.
4. Check unground flats for being central, observing which side is high and whether or not it is within the grind stock, as shown in Fig. 769(c).
5. Place the job on the chuck with the low side up, and spot the wheel to clean up

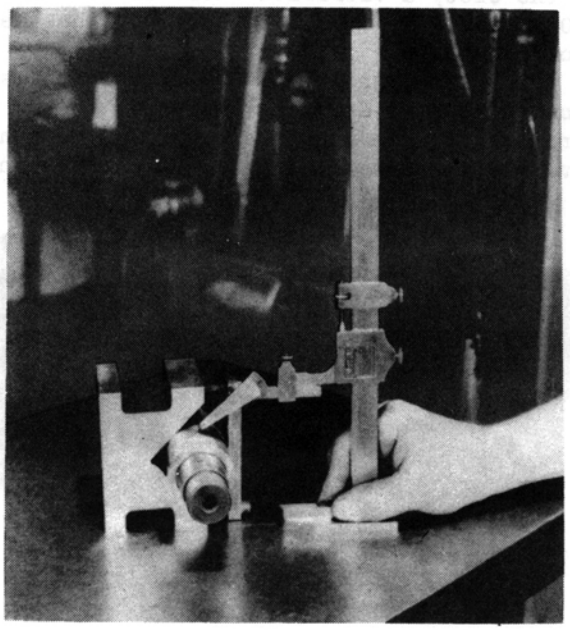


Fig. 769(b)

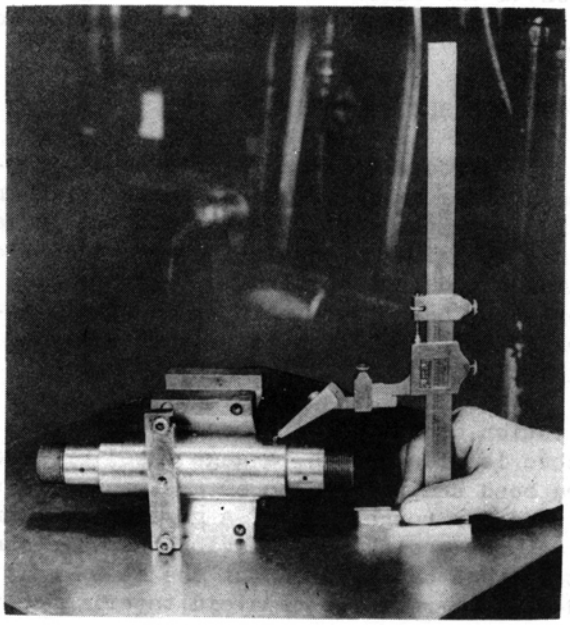


Fig. 769(c)

this side of the flat leaving witness marks.

6. Reverse the V-block to bring the unground side of the job up, and take the same cut from this face as shown in Fig. 769(d).

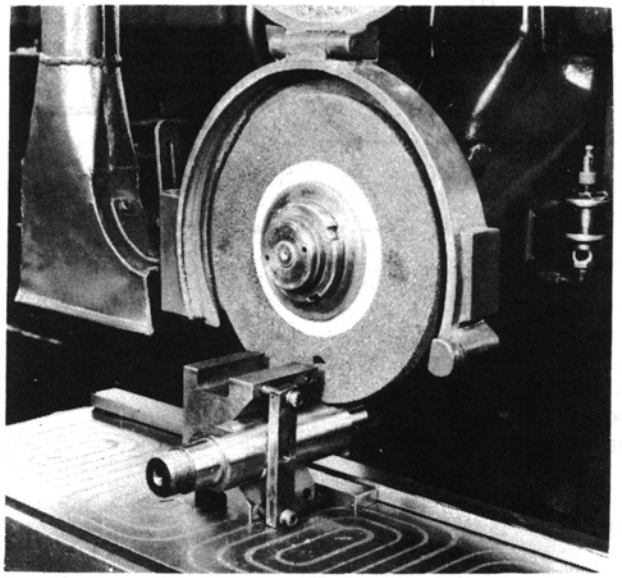


Fig. 769(d)

7. Check the job for size noting by what amount it is oversize.
8. Feed the grinding wheel down an amount equal to one-half the remaining stock and grind both faces; reversing V-block as in step 6. This should bring the job to the required size.
9. Check the flats for size and being central; then have it checked by the inspector.

GROOVE AND SLOT GRINDING

Certain classes of work on the surface grinder call for considerable skill and patience on the part of the operator. Some of the more common ones being groove or slot grinding and radius grinding, especially where they have to be maintained within accurate limits relative to another surface (Fig. 770).

Probably the most important factor is selection of the proper wheel. Where the groove is wide enough, a gage wheel of medium hard bond and from 46 to 60 grain size should be used for roughing. If a gage wheel cannot be secured a disk wheel having the same characteristics can be used but the sides must be undercut or relieved to avoid tapering on the top of the slot or groove.

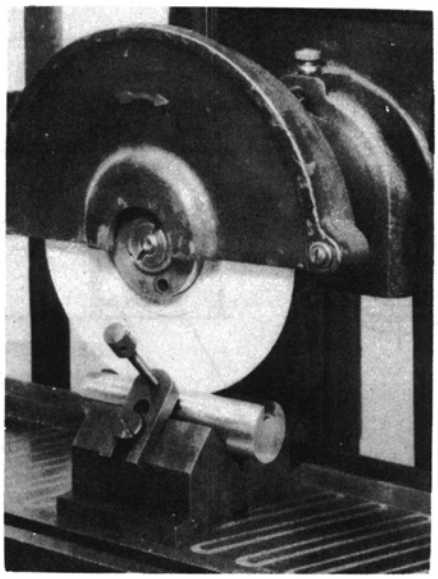


Fig. 770

"Bell mouthing" of a slot or groove is a condition where the ends of the slot become gradually wider than the center. On the surface grinder this condition is generally caused by spindle end play or pressure on the wheel; the work forces the end play or wheel in one direction, then as the wheel clears the slot the work pressure decreases permitting the spindle to occupy a normal running position. This condition can be overcome by doing most of the grinding in the central section of the groove or slot and only occasionally running the wheel off the ends.

To grind sharp corners on the bottom of the slot, a vitrified wheel of 80-0 or 120-P should be used after the slot has been ground to size.

Form grinding can be done on the surface grinder by dressing the wheel to the desired shape. In the case of convex or concave radii a special diamond holder, known as

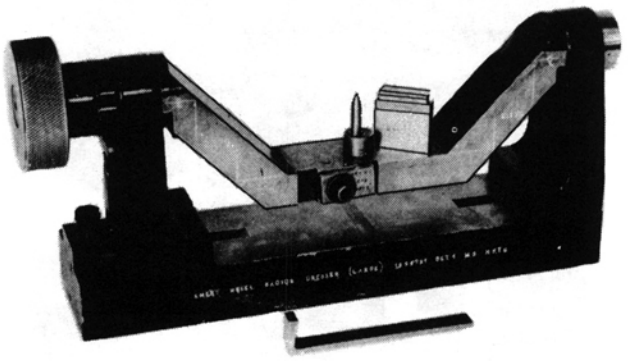


Fig. 771

a radius cradle, (Fig. 771) permits the diamond to be accurately set to the required radius.

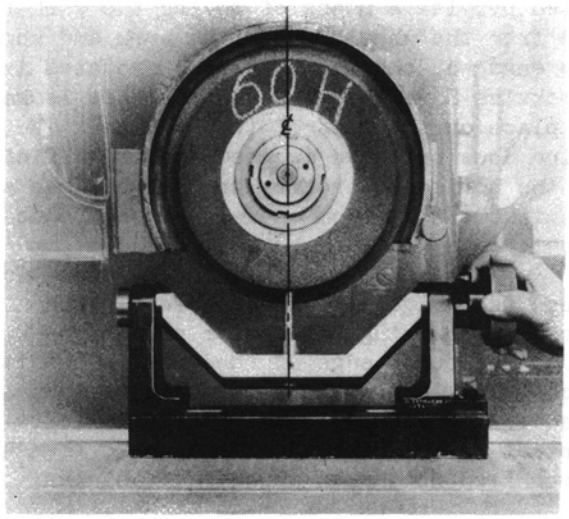


Fig. 772

TO USE THE RADIUS CRADLE

Suppose that it is necessary to grind a 3/8" convex (male) radius. The wheel would have to be dressed concave as shown in Fig. 773. Adding 3/8" to "C" gives the distance which the diamond point is to be located above the gaging surface of the cradle; "C" being the distance from the axis of rotation to the gaging surface of the cradle.

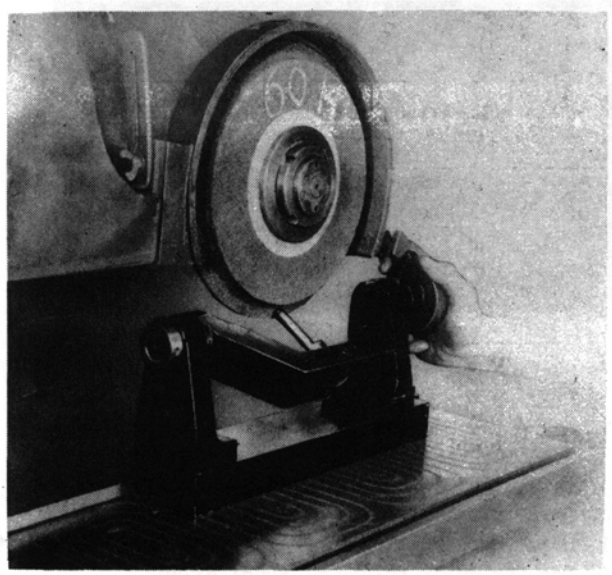


Fig. 773

Stack "Jo" blocks to this distance, place the blocks on the gaging surface and adjust the diamond gage to the blocks and lock it in place; insert the diamond holder in the cradle and adjust it to the diamond gage, lock it in place then remove the diamond gage and the cradle is ready to use. The same procedure is followed for setting the diamond to dress the wheel convex (Fig. 774), except that the radius is subtracted from "C".

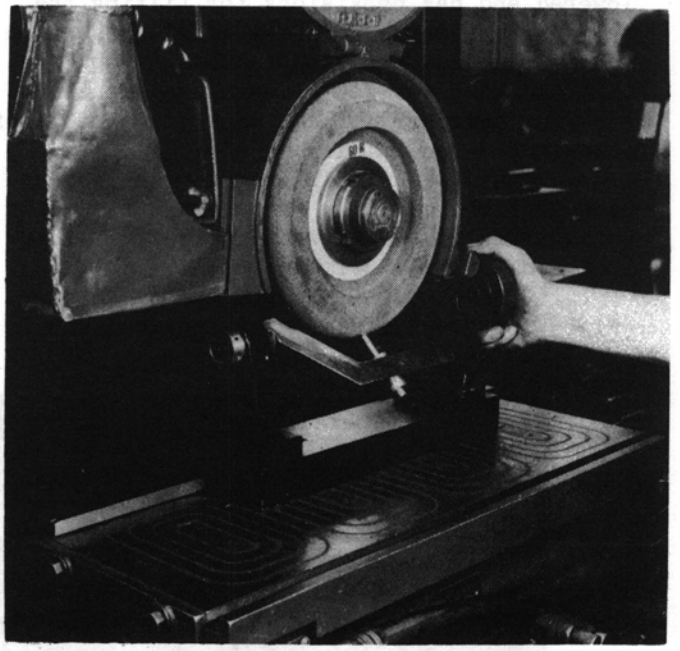


Fig. 774

It is very important when dressing the wheel for an angle or radius to have the diamond point set on the vertical centerline of the wheel as shown in Fig. 772, otherwise the wheel will be dressed to the wrong form.

The table of the horizontal planer surface grinder traverses very easily, and may be pushed back and forth by leaning against the end of it. This is a desirable quality of the table in so far as surface finish is concerned, but becomes dangerous when removing work from the magnetic chuck. Many ugly hand wounds are suffered by operators of this machine because they move the table to the right of the wheel, then take hold of the work and give it a pull in the direction of the wheel, so that if the table moves or when the work lets go the operator's hand or arm comes in contact with the revolving wheel resulting in a nasty burn and cut.

AFTER THE WORK TABLE HAS BEEN MOVED TO THE RIGHT OF THE WHEEL AND THE MAGNETIC CHUCK TURNED OFF, THE OPERATOR SHOULD REMOVE THE WORK FROM THE CHUCK BY PULLING IT IN A

DIRECTION PERPENDICULAR TO THE LONGITUDINAL TRAVEL OF THE TABLE

The HORIZONTAL ROTARY grinder (Fig. 775) commonly known as a "Ring grinder" consists of a horizontal wheel spindle having a reciprocating motion similar to that of the shaper ram and a revolving magnetic chuck table supported by columns at the front of the machine. The work table can be raised or lowered and has provisions for tilting the table for concave or convex grinding. The machine is equipped with a coolant supply tank and pump for wet grinding and since it uses the periphery of the wheel is capable of producing a good finish

This machine is used for the production of flat, concave, or convex surfaces which makes it readily adaptable for either tool room or production purposes.

The VERTICAL SPINDLE ROTARY grinder (Fig. 776), consists of a cylindrical wheel mounted on a vertical spindle and supported on a vertical column. This vertical column provides a means of raising or lowering the wheel. The work table consists of a revolving magnetic chuck supported on ways or slides, which provide a means of moving the work to and from the wheel. When the work and wheel are engaged, the magnetic chuck rotates in a clockwise direction but the table is locked in place on the ways of the machine. The machine does not give as good a surface finish as the horizontal rotary grinder but has a use in the tool room as well as on production.

The work is placed on the magnetic chuck (see Fig. 777), the table is then moved in to bring the center of the chuck under the outer edge of the wheel where it is locked in place. The wheel head is then lowered very gradually until sparks indicate contact between the wheel and work, then the power feed is thrown in. If one of the

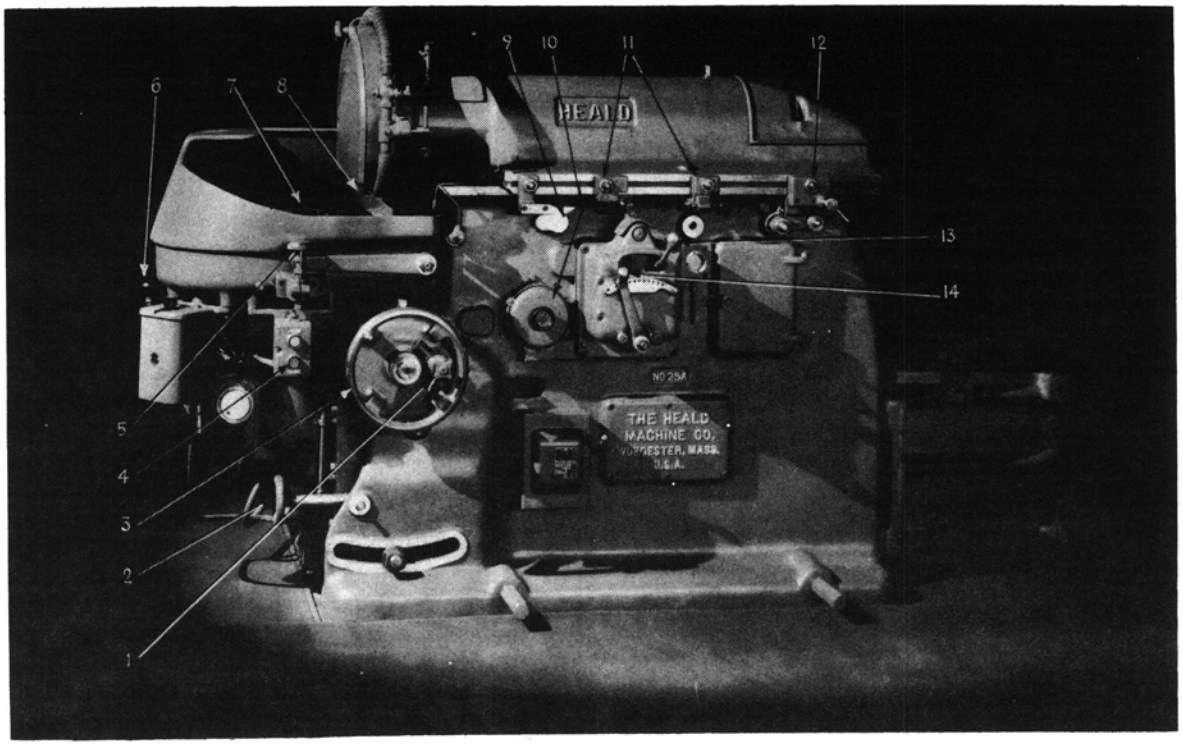


Fig. 775. Horizontal Rotary Surface Grinder

1. Knob for fine adjustment of work to wheel.
2. Handwheel for adjusting chuck to give concave and convex surfaces.
3. Handwheel for raising and lowering work chuck.
4. Start and stop buttons for rotation of magnetic chuck.
5. Diamond mechanism for truing wheel.
6. Off and on switch for magnetism to chuck.
7. Magnetic chuck for holding work.
8. Grinding wheel.
9. Safety dog for wheel slide.
10. Lever for adjusting speed of wheel slide.
11. Dogs for adjusting length of stroke of wheel slide.
12. Dog for quick return of wheel slide.
13. Wheel slide reverse lever.
14. Lever for adjusting speed of wheel slide and wheel.

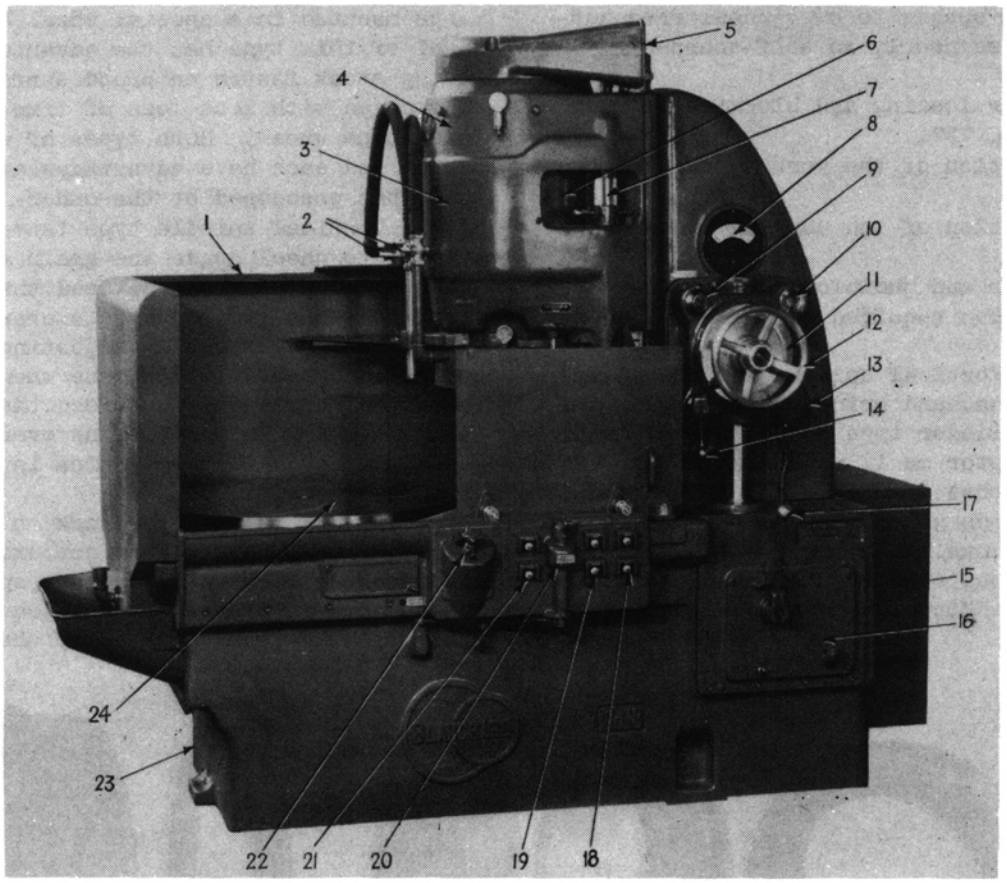


Fig. 776. Vertical Spindle Rotary Grinder

1. Heavy Welded Steel Guards
2. Water Cocks
3. 25 H.P. Induction Motor
4. Ball Bearing Wheel Head
5. Air Outlet
6. Air Inlet
7. Wheel Dresser
8. Ammeter
9. Feed Variator
10. Oil Flow Indicator
11. Feed Dial 1 Turn = .100"
12. Feed Wheel 1 Turn = .025"
13. Oil Filter
14. Feed and Head Elevating
15. Control Cabinet
16. Chuck Speed Box With Oil Pump
17. Chuck Speed Control
18. Pump Control
19. Wheel Control
20. Table Traverse Control
21. Chuck Rotation Control
22. Chuck Switch
23. Base
24. One Piece Steel Magnetic Chuck

pieces has been previously ground to size at one spot and coated with blue vitriol and placed on the chuck, it can be used as a sizing block. The wheel can then be fed into the work until light scratches appear on the vitriolized surface indicating that the work is to size. When the correct size is reached the down feed is stopped, the wheel head raised, the work table moved out and the job removed.

The more essential points for the efficient operation of a Blanchard grinder are given below:

1. The selection of the proper grade and grain of wheel for the job.

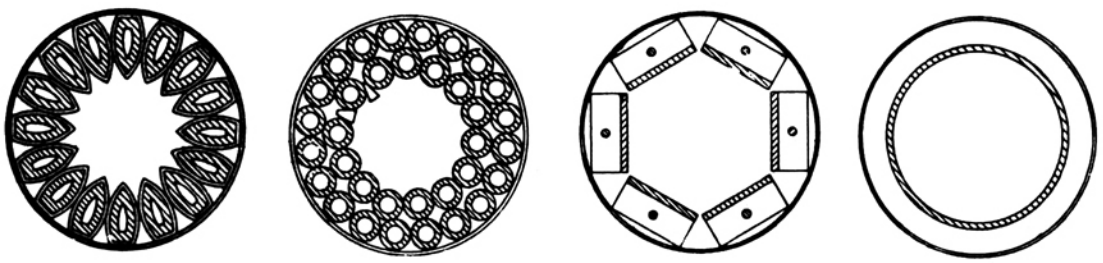


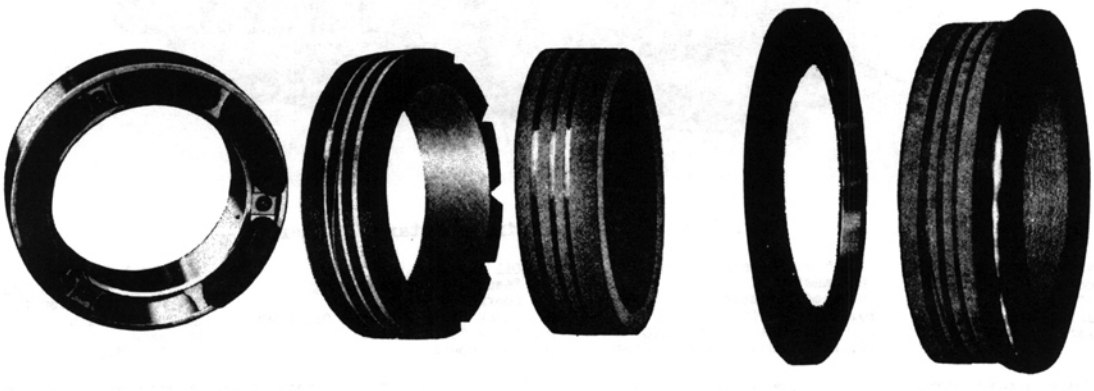
Fig. 777. Methods of Placing Various Types of Work on the Magnetic Chucks of Vertical Spindle Grinding Machines.

2. The selection of the proper wheel feeds and chuck speeds to keep wheel free cutting and as nearly as self-sharpening as possible.
3. The proper loading and blocking of the work, Fig. 777.
4. The condition of the working face of the chuck.
5. The condition of the working faces of the work.
6. The proper and judicious use of the wheel dresser when required.

The Vertical spindle surface grinder, such as a Blanchard grinder, etc., ordinarily uses a cylinder type wheel mounted in a suitable adaptor as illustrated in Fig. 778. Take notice that this wheel is reinforced with wire bands which must be removed as the wheel wears down.

Another type of wheel used on a machine of this kind is known as a segment wheel. It consists of a number of wheel segments mounted in a special wheel chuck. A wheel of this type has the advantage of removing stock faster on broad surfaces and may be changed with less loss of time than a cylinder type wheel. Both types of wheels are useful and each have advantages and disadvantages not possessed by the other. A machine of the Vertical spindle type is very sensitive as to wheel grade and grain and it is important that a wheel be used that is free cutting. Much may be done to overcome glazing of the wheel face by adjusting the speed of the chuck rotation and the wheel feed; it is better to slightly overfeed than to underfeed the wheel to the work as overfeeding tends to keep new sharp grains in contact with the job.

Where the excess stock on a large number of pieces of work does not exceed one-eighth of an inch, the vertical spindle grinder does a much faster job and leaves a much better finish than the planer, shaper, or mill.



(a) (b) (c) (d) (e)

Fig. 778. (a) 18-in. Segment Wheel; (b) 18-in. Sectored Wheel; (c) 18-in. Cylinder Wheel; (d) Ring Adaptor; (e) Wheel Cemented in Ring Adaptor.

TOOLS COMMONLY USED ON SURFACE GRINDERS

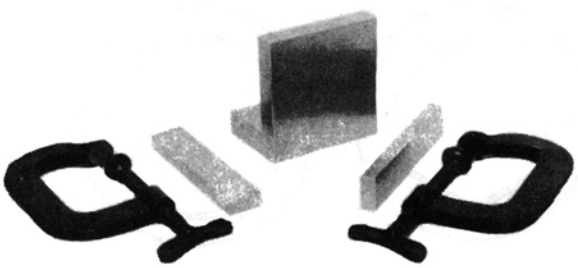


Fig. 779. 3" Angle Plate; H. & G. Parallels; 3" "C" Clamps.

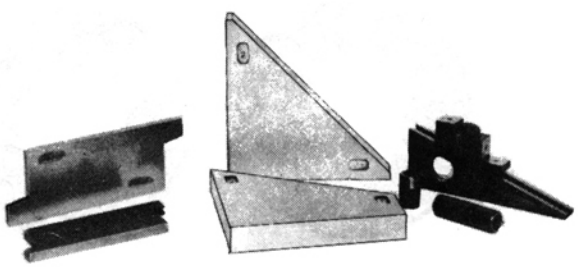


Fig. 780. (left) Master Angle Blocks; (center) Planer Gage; (right) Adjustable Parallels.

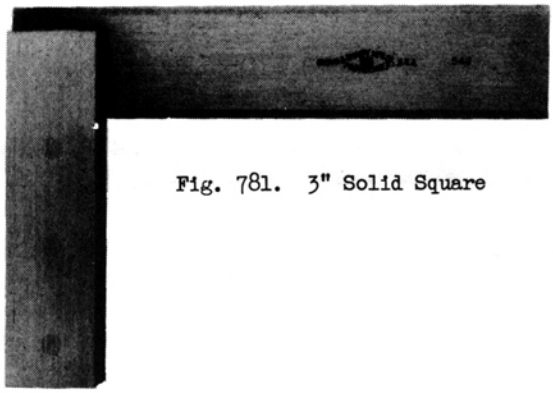


Fig. 781. 3" Solid Square

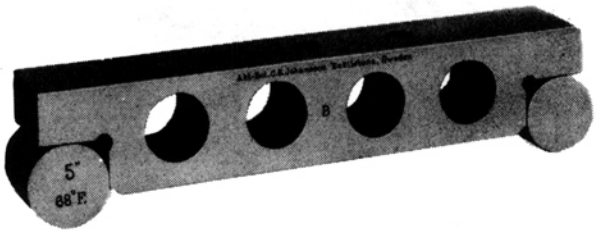


Fig. 782. 5" Jo Sine Bar

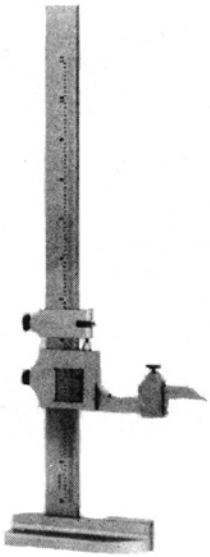


Fig. 783. 10" Height Gage

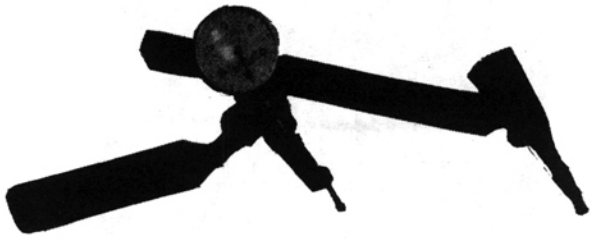


Fig. 784. Last Word and Ideal Indicator

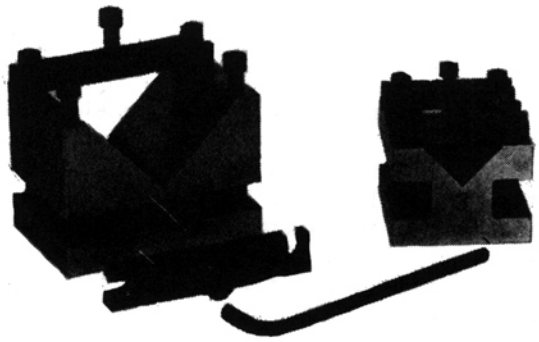


Fig. 785. 3" V-Blocks

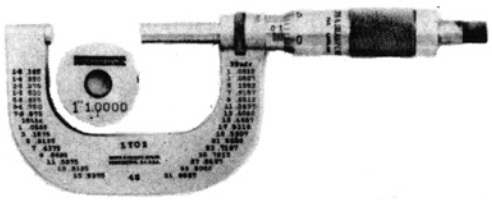


Fig. 786. 1-2" Mics

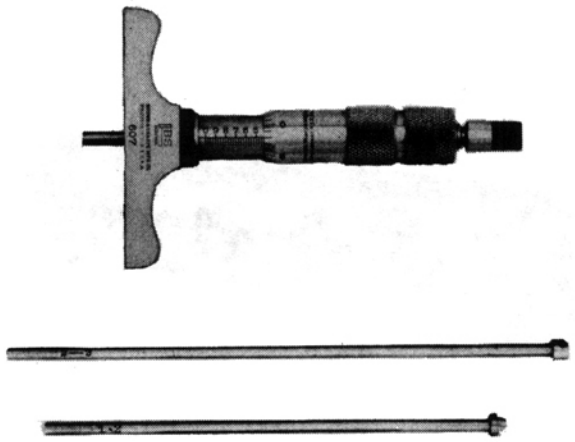


Fig. 787. Depth Mic.

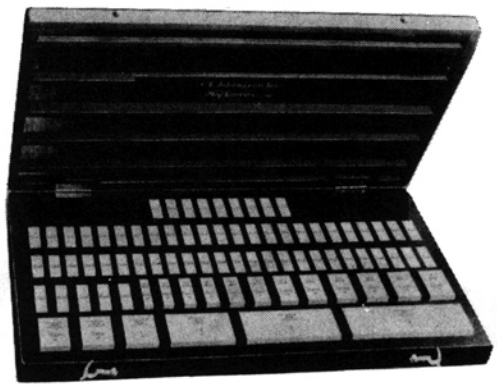


Fig. 789. Johansson Gage Blocks

SAFETY RULES FOR SURFACE GRINDERS

1. Test all wheels for cracks or defects before installing them on the spindle.
2. When starting a machine with a new wheel, stand safely to one side until you are sure that the wheel is sound.
3. Whenever there is danger of injury in any way to yourself, in loading, unloading, or checking work on the chuck, STOP the machine.
4. In removing work from the magnetic chuck, crank the table clear of the grinding wheel and then pull the stock AWAY from the wheel.
5. Small thin pieces of stock, extra long stock, or stock with small contact surfaces should be thoroughly blocked on the chuck.
6. Do not use an unguarded wheel at any time if it has a diameter greater than 1-1/2 inches.
7. Wear goggles when using a rubber wheel on a surface grinder.
8. Do not wipe a chuck with a towel--use an oil brush.

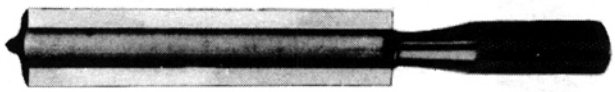


Fig. 788. Johansson Straight Edge

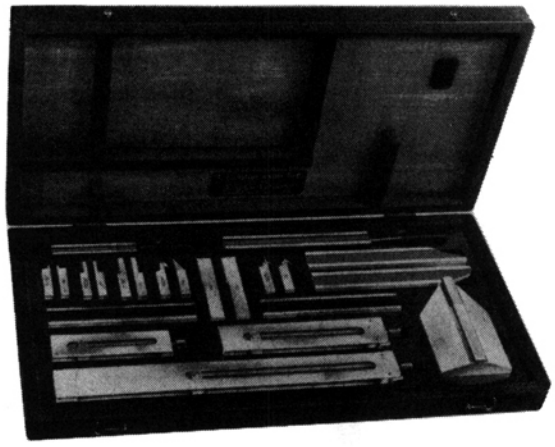


Fig. 790. Johansson Accessory Set

9. Shut off the machine while checking or setting up a job.
10. Use properly undercut safety washers in mounting a wheel.
11. Exhaust hoods are supplied as a safeguard for the health of grinder operators. See that they are properly adjusted at all times and that they are not abused.

The one and two inch micrometer and solid square are the most common precision tools used on the surface grinder. The 10" height gage and Ideal or Last Word indicator is frequently used in connection with a planer gage or gage blocks. The planer gage and adjustable parallels are frequently used for measuring the widths of slots instead of the vernier caliper because they are easier to set and give a more positive measurement. A straight edge is frequently used on the surface grinder to test the flatness or straightness of a surface.

QUESTIONS

1. What is meant by surface grinding?

2. What is the difference between the planer and rotary types of surface grinder?
3. What companies manufacture the surface grinders used in the Henry Ford Trade School?
4. Name the principal parts of the surface grinder shown in Fig. 754.
5. What wheel shapes are commonly used on the surface grinder?
6. What are the grain and grade sizes of grinding wheels commonly used on surface grinders?
7. How should the wheel be inspected for cracks?
8. What is the purpose of blotting paper gaskets on the sides of the wheel?
9. What is meant by "safety flanges"?
10. Why is it necessary to step to one side when starting the machine up after it has stood idle all night or after a new wheel has been installed?
11. Why is it necessary to keep surface grinder wheels sharp and clean?
12. What is the result of continued use of a dull or dirty wheel?
13. Give five common methods of holding work on the surface grinder.
14. Explain the correct method of holding small work on a surface grinder?
15. How are non-magnetic materials held on the magnetic chuck?
16. What is the purpose of the "back rail" on the magnetic chuck?
17. What is the purpose of a 90° angle plate?
18. In what condition should the angle plate be kept for accurate work?
19. How many faces can be ground square with each other at the same time by use of the 90° angle plate?
20. From Figs. 761 and 762 explain how to set up a job for squaring by use of an angle plate.
21. Give two reasons for work being out of parallel in surface grinding.
22. Give an efficient method for keeping thick work parallel on the surface grinder.
23. How is thin work prevented from warping on the surface grinder?
24. What is meant by grinding the work to "clean up"?
25. Explain what "witness marks" are.
26. Briefly explain how to surface grind a job to leave witness marks on it.
27. When is the V-block used for holding work on the surface grinder?
28. Name four factors governing the accuracy of a V-block.
29. Briefly explain the procedure in grinding two opposite flats central on a cylindrical shaft.
30. Give two methods for grinding angular work.
31. Why should the grinding wheel NOT be dressed at an angle unless absolutely necessary?
32. When dressing the wheel at an angle what is an important thing to remember?
33. How is the wheel dressed at the desired angle?
34. What tools are commonly used for setting the work at an angle?
35. What is a "sine" bar?
36. Explain how the sine bar is used to set the work at an angle.
37. What two surface grinding operations call for considerable skill and patience on the part of the operator?
38. What shape, grain size, and bond of wheel should be used for slot grinding?
39. How can an ordinary disk wheel be used for grinding?
40. What is meant by "bell-mouthing" of the slot?
41. How can bell-mouthing of the slot be prevented?
42. How is it possible to maintain sharp corners in the bottom of a slot?
43. How is it possible to grind radii on the surface grinder?
44. What is the purpose of a radius cradle?
45. Explain briefly how a radius cradle is used.
46. What is the important thing to remember when dressing a radius or angle on a wheel?
47. What is the correct method of removing work from the magnetic chuck of a surface grinder?
48. What machine is frequently referred to as a "ring" grinder?
49. What is the principle of operation of the ring grinder?
50. What work operations can be performed on the ring grinder?
51. What is the principle of operation of the vertical spindle rotary surface grinder?
52. Describe the work table of the vertical spindle rotary surface grinder and tell how it operates.
53. After the work is loaded on the magnetic chuck how is it brought into contact with the grinding wheel?
54. How is sizing of the work accomplished on the vertical spindle rotary surface grinder?
55. Give six important points for the efficient operation of a vertical spindle rotary surface grinder.
56. What types of wheels are used on the vertical spindle rotary surface grinder?
57. How may glazing of the wheel be prevented on the vertical spindle rotary surface grinder?

58. What is safety rule #6 for surface grinders?
59. What is the purpose of exhaust hoods on grinders?
60. What precision tools are commonly used for checking work on the surface grinders?

INTERNAL GRINDING

Internal grinding is the act of grinding straight cylindrical, tapered or formed holes to accurate size. It is done on the Universal grinder (Figs. 791 and 792) or on machines especially designed for that purpose. Fig. 793 shows a collection of work commonly done on the internal grinders.

Machines for doing internal grinding are divided into three groups depending on how the work is held and operated. The Work Rotating type uses a three or four jaw chuck,

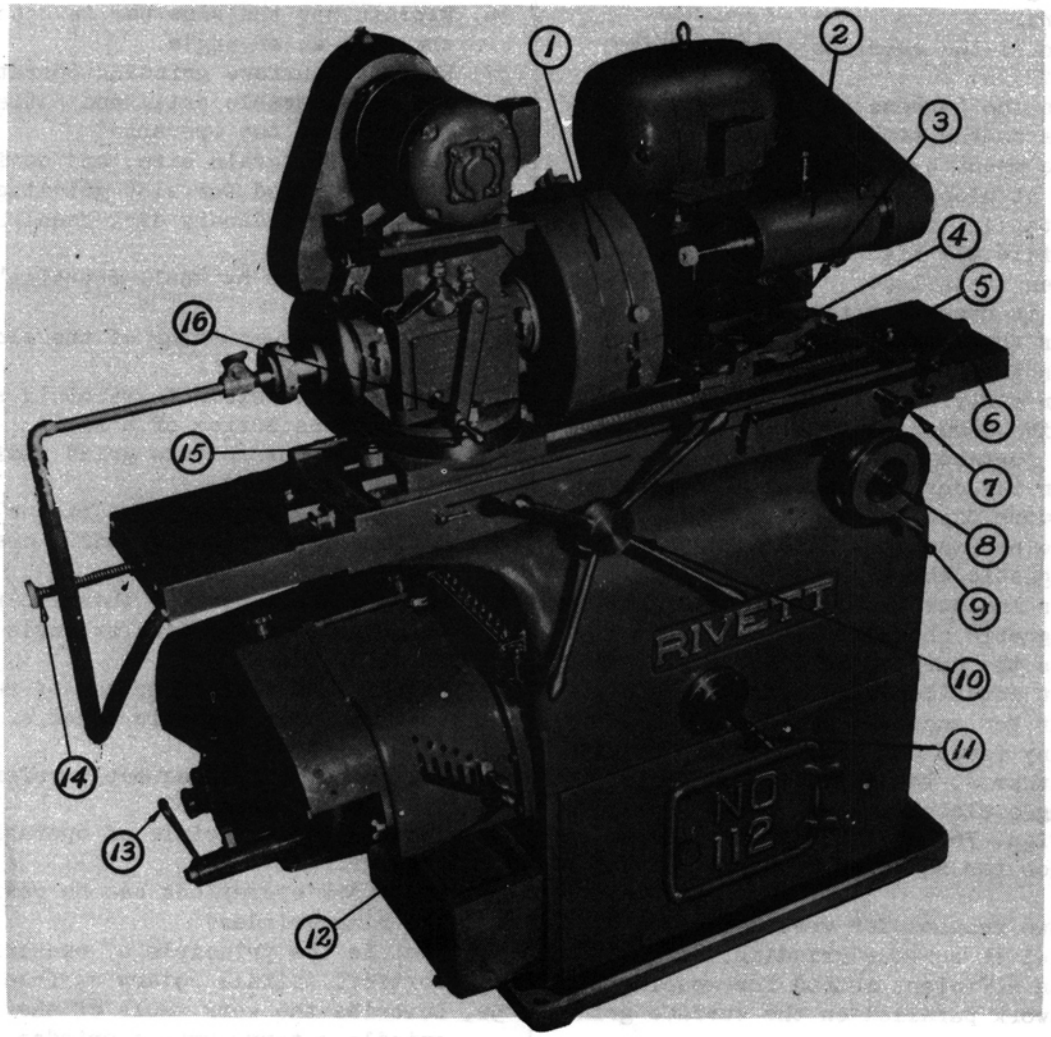


Fig. 791. Universal Internal Grinder

1. Guard. Covers spindle nose and hinges up for access to chuck, faceplate, or collet.
2. Wheel head on graduated swiveling base.
3. Wheel head slide, moves in and out.
4. Fixture for tripping diamond truing device.
5. Fine adjustment for upper table swivel.
6. Table traverse locking lever.
7. Screw for adjusting upper table swivel.
8. Knob for engaging wheel head slide power cross feed.
9. Handwheel for operating wheel head slide. Is graduated in .0005".
10. Pilot wheel for traversing table by hand.
11. Crank for setting length of stroke of table.
12. Change gear handle for changing speed of table traverse.
13. Belt tension adjusting lever.
14. Screw for positioning table stroke.
15. Work head base. Graduated in degrees for cutting steep tapers.
16. Lever for starting and stopping work head.

face plate or special fixture for holding the work. The Centerless internal uses a set of rollers which hold the work and give it a revolving motion. The Cylinder grinder holds the work in a fixed, non-rotating position on a reciprocating table and depends on the amount of eccentric wheel spindle travel to generate the correct size of the hole.

THE WORK ROTATING TYPE is the one commonly used in tool and die rooms. A three or four jaw chuck or drive plate screws on the spindle nose of the work head. The work head is mounted on the work table which in some cases moves back and forth. On others the wheel head moves back and forth, the work table being in a fixed position.

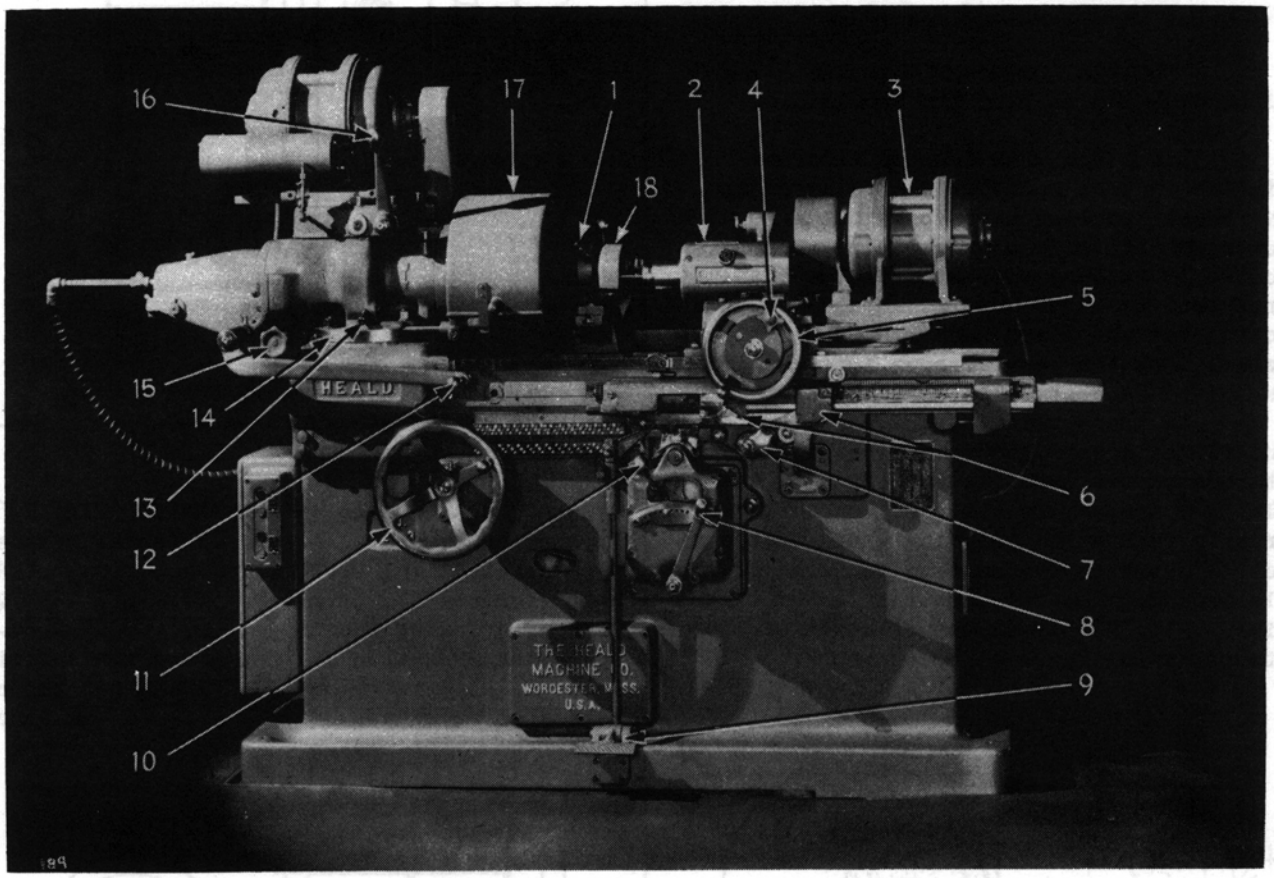


Fig. 792. Plain Internal Grinder

1. Diamond for truing wheel.
2. Grinding wheel head.
3. Motor for driving wheel head.
4. Fine adjustment for feeding wheel to work.
5. Cross slide hand wheel for feeding wheel into work.
6. Dogs for controlling length of stroke of table and grinding wheel.
7. Knob for adjusting engagement of quick return of wheel head.
8. Table speed control lever adjusts from 0 to 44 F.P.M.
9. Foot treadle lifts dog and lets table and wheel head go to rest position.
10. Ball lever reverses table in direction it is thrown.
11. Hand wheel for controlling movement of the table.
12. Lever operates collet for holding work.
13. Spindle lock knob convenient when removing fixtures.
14. Plate graduated for setting work head to grind tapers.
15. Knob for swiveling and lining up work head.
16. Lever for starting and stopping work head.
17. Guard around chuck and spindle nose. Lifts up.
18. Grinding wheel guard. Works automatically.

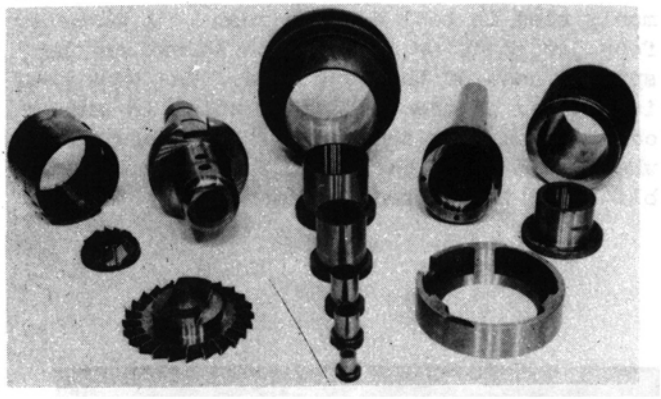


Fig. 793. Samples of Work Done on the Internal Grinder in Henry Ford Trade School.

Since the work done on internal grinders in tool rooms is generally ground dry, these machines have to be built to much closer specifications than other grinders, because it is harder to protect the vital parts from the ever present abrasive dust and because the wheel and work speeds are much faster. For the same reasons internal grinders should be kept well lubricated.

In these grinders the wheel and work centerlines are in the same horizontal plane, but because the wheel is smaller than the hole they are not in the same vertical plane; for this reason the grinding wheel must contact the work on the near or far side of the hole depending on the construction of the machine. Fig. 794 shows the relation of the internal grinding wheel to the work.

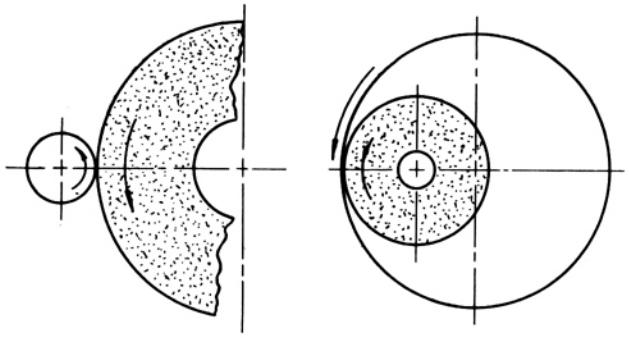


Fig. 794. (left) External; (right) Internal

In chucking a job in a four jaw chuck, care must be exercised to get the work running true. Fig. 795 shows a job in the chuck and the points at which it must be indicated to get it running true. The dial indicator is placed first at point "A" and the chuck jaws adjusted to give a minimum of run-out to the work at that point; then the dial indicator is moved to point "B". At this point the work must be forced into line by tapping it with a mallet. This procedure should be repeated until the job runs true within the required limits.

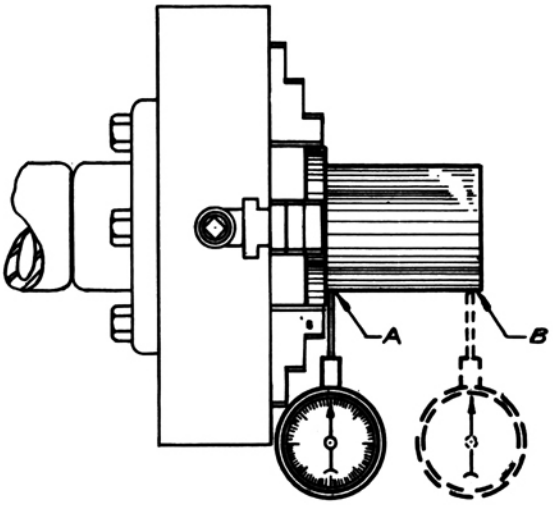


Fig. 795

The following outline is a typical analysis of the procedure in internal grinding.

Type of job . Bushing as per sketch (Fig. 796)
Type of machine . . . . . .Internal grinder
Type of steel . . . . . ."RR"
Heat treatment . . . . . "Q"
Kind of grinding wheel. 60-K
Operations required . . Rough and finish grind ID as per sketch
Method of holding . . . . . 4 jaw chuck

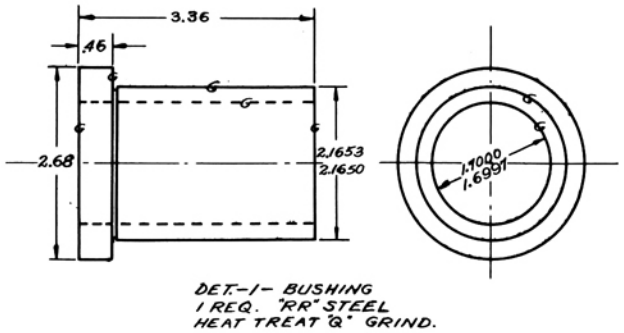


Fig. 796

PROCEDURE

1. Check ID for sufficient grind stock.
2. Mount a four jaw chuck on head stock and chuck up the work according to the discussion on chucking.

3. Select the proper quill and grinding wheel, mount them on the wheel head and dress the wheel. The proper quill to use is one which is as short and strong as possible, consistant with the length and diameter of hole to be ground.
4. Adjust the machine for length of stroke and set the table to grind straight.
5. Take a trial clean up cut removing as little stock as possible.
6. Check the hole for straightness and make any necessary table adjustments.
7. With the wheel cutting straight, rough out the hole to within .001" or .002" of the required size.
8. Dress the wheel for finishing and finish grind the hole to size.
9. Check the hole with the proper size gage, noting any bell-mouthing or out of roundness. Have the first piece inspected.

METHODS OF HOLDING WORK

Besides holding work in a three or four jaw chuck, it may also be held in a collet; by clamping it directly to a face plate; by holding the work in a V-block and clamping the V-block to a face plate; by clamping the work to a 90° angle plate then clamping the angle plate to a face plate; or in case of a large number of pieces of the same kind it may be held in special fixtures or adaptors. Work may also be held by a chuck and a steady rest as shown in Fig. 797. At times it is necessary to grind the hole in a spindle like piece of work which necessitates the use of a steady rest, and tying the end of the job to the headstock center and drive plate. Fig. 798 shows such a set up. Page 259 illustrates a series of more common set ups and operations as performed on an internal grinder.

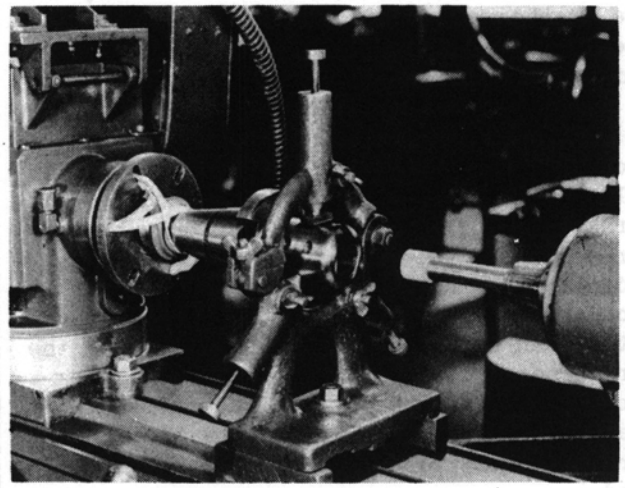


Fig. 798

When holding thin-walled bushings in a chuck the jaws should be tightened only sufficient to hold the work, otherwise the pressure exerted by the jaws will spring and distort the hole. In like manner when clamping work to a face plate, be sure that the work is firmly seated and that the pressure is evenly distributed and only sufficient to hold the work firmly.

Fig. 799 shows the correct and incorrect methods of clamping work to the face plate. On the right-hand side note that the clamp is not parallel to the face plate because the rest block is too high; note that the bolt is too far away from the job; note that the parallel, upon which the job depends for end clearance of the grinding wheel, is set too far in, and will interfere with the grinding operation. The bolt, clamp, rest

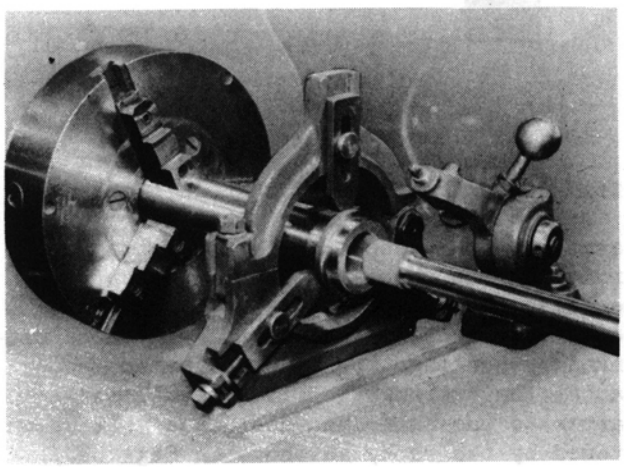


Fig. 797

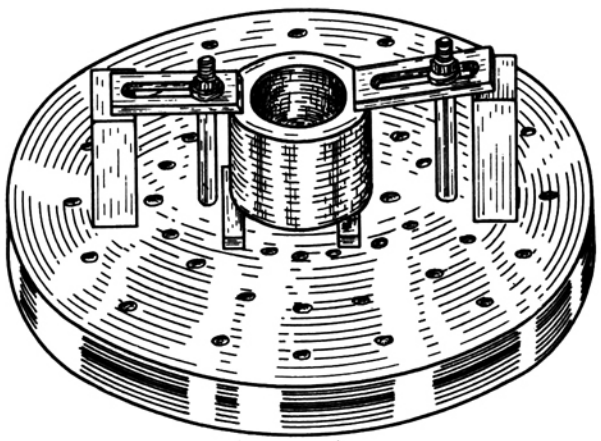


Fig. 799

block, and parallel are correct on the left-hand side.

Out-of-roundness of the work may be due to the work being improperly supported, overheating of the work, loose work head spindle, improper clamping, etc.

Generally speaking, the operator of an internal grinder is required to do one of two things, that is either internal cylindrical grinding or internal taper grinding. Internal cylindrical grinding can only be performed if the wheel and work heads are coordinated so that the axis of rotation of each are moved in parallel planes. This condition can be brought about by adjusting the work head on its swiveled base or the work table so that the horizontal centerline of the work will be parallel to the back and forth movement of the work table or wheel head.

EXTREME TAPER is produced in the work by swiveling the work head to one-half the taper angle by means of the graduations on the circular base of the head. Slight tapers may be produced by adjusting the table to the correct T.P.F. These graduations are obviously not accurate enough for precision work, so that a taper plug gage of the correct taper per foot is used for checking the work after a clean up cut is taken. The plug gage is given a light coat of Prussian blue and with a twisting motion is inserted in the hole to be checked. If the two tapers are not identical, a bright metallic line or surface will show on the plug gage which will indicate whether the taper is too much or not enough. If the head has been swiveled too much the brightened surface will appear on the small end of the plug gage. If the head has not been swiveled enough the brightened surface will appear on the large end of the plug gage.

When using plug gages for checking work be careful that the plug doesn't "freeze" in the hole. "Freeze" is the term applied to a condition where the plug is held fast by the work and is brought about by the fact that the heat of grinding causes the work to expand, so that when a cold plug gage is inserted, the work contracts thus locking the plug gage in the work.

Where a number of pieces are being ground this freezing may be prevented by keeping the plug and the work at the same approximate temperature. Leave the plug gage in the hole of the last piece ground until it is necessary to check the hole in the next piece. This keeps the plug gage warm and helps to prevent freezing.

The diameter of the wheel for internal grinding is based on securing the stoutest possible quill for maximum support, consistent with the size of the hole. Generally speaking the diameter of the wheel should not exceed two-thirds of the diameter of the hole. It must be remembered that as the size of the wheel increases, the diameter of the hole being constant, the greater the area of contact becomes between the wheel and work thus increasing the heating and probable distortion of the work.

The manufacturers of internal grinding machines build the limits of the wheel speed into the machine, and are such that speeds from 4000 to 6500 S.F.P.M. can be obtained. These speeds, of course, depend on the size of the wheel.

Various kinds of diamond or tungsten carbide dressers are available for mechanically or hand dressing the wheel. Occasionally grinding wheels are dressed by the operator holding a piece of silicon carbide in his hand, and passing it along the periphery of the wheel so that the wheel will contact the work properly.

The amount of stock to be removed from a hole to bring it up to size, depends on the diameter and length of the hole but is generally from .004" to .012". More stock than this, means a longer time to complete the job since the hole is smaller, and greater pressure and lighter cuts have to be taken.

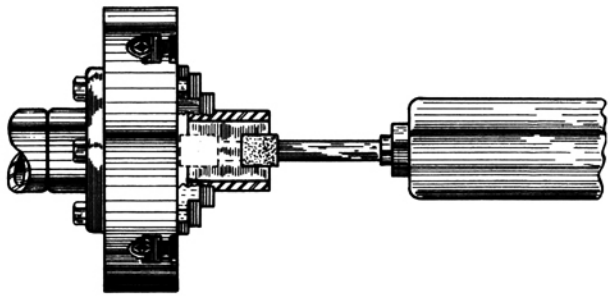


Fig. 800

"BELL-MOUTHING" of the work is a condition where the ends of the hole flare out or are increasingly larger than the required diameter. This condition is brought about by permitting the wheel to overrun the ends of the hole, wheel too hard, excessive wheel pressure, or wheels too short. It can be prevented by setting the length of stroke so that at the extremity of its travel, one-quarter to one-half the wheel face will be uncovered by the work, as shown in Fig. 800, and correcting other factors as listed above.

The work should be checked for size

and straightness after a clean-up cut has been taken, but be sure to move the work and wheel far enough apart to enable the wheel guard to swing down and cover the wheel, (Fig. 801) otherwise serious cuts and burns are apt to be received from the running wheel.

Wheel guard swings downward to protect operator, but does not interfere with fixtures. Can be adjusted for all sizes of wheelheads

Fig. 801

BLIND HOLES can be ground on the internal grinder provided an undercut of sufficient width is made to give the end of the wheel a little over-run of the hole.

The INTERNAL CENTERLESS, Fig. 802 is especially designed for big runs of work of the same kind. It is fully automatic and assures that the hole will be concentric with the outside diameter of the work. It is capable of producing straight or taper, blind or through holes, interrupted holes and holes having a shoulder.

In this machine the work is driven and its speed controlled by a regulating roller which causes the work to revolve in the same direction as the grinding wheel. To support the work a second roller is mounted below the work and may be adjusted for varying distances from the work center. A third roller, known as a pressure roll, is supported on a swing bracket and holds the work in contact with the other two rollers and moves in and out to allow for loading and unloading the machine. The work head moves back and forth while the wheel head is fixed longitudinally. This makes for greater rigidity for the wheel, which keeps vibration reduced to a minimum.

The CYLINDER GRINDER, Fig. 803 commonly known as a "Planetary Grinder," was designed for the expressed purpose of doing internal grinding on work which was too cumbersome to be revolved on a work head. It is used to a good advantage in railroad shops or shops engaging in heavy machinery operation.

This machine has a reciprocating table to which the work is clamped in a fixed position. The grinding wheel is mounted on a spindle which, while revolving, travels in an adjustable circular path, the size of the path determining the diameter of the hole.

All of the internal grinders considered here, use Vitrified wheels of from 1/4" to 2-1/2" in diameter and from 3/8" to 2" face. The more common grades and grain sizes of aluminous oxide wheels used for internal grinding are; 46-I, 60-I, 46-J, 60-J, 60-L, and 120-P.

SAFETY RULES FOR INTERNAL GRINDING

1. Internal grinding wheels are provided with safety guards that enclose the wheel while the work is being checked. See that this guard is working at all times. Do not work without it in operation.
2. Mount all internal grinding wheels with either paper or brass washers; brass preferred for small wheels.
3. See that the machine is shut down when setting up a job, or while making any necessary changes to guards, stops, or other machine attachments.
4. Wheel breakage during facing operations is large. Keep the wheel dressed sharply and use goggles when doing work of this nature.
5. Do not use wheels in excess of 2-1/2 inches diameter on the internal grinder without a special permit from the instructor.
6. Do not try to wipe out the hole while the work head is in motion. This is a bad practice resulting in finger and hand injuries.
7. See that all heavy work is properly and securely clamped; use standard planer jacks and grinder clamps. Work off-center should be counterbalanced.
8. When using thin rubber wheels or extra hard wheels on the internal grinder, wear goggles as an extra protection from wheel breakage.
9. Any class of work that is too long to allow the proper use of the safety wheel guard while checking the work is a source of danger. Shut the machine down to check work of this nature.

QUESTIONS

1. What is meant by "Internal" grinding?
2. On what kinds of machines is internal grinding done?
3. What companies manufacture the internal grinders used in Henry Ford Trade School?
4. Into what three groups are internal grinders divided?
5. What kind of internal grinders are commonly used for tool room work?
6. How is work commonly held in the work rotating type of internal grinder?
7. Name the parts of the internal grinder which are pertinent to its operation.
8. Why must the internal grinder be built to closer specifications than most other machines?

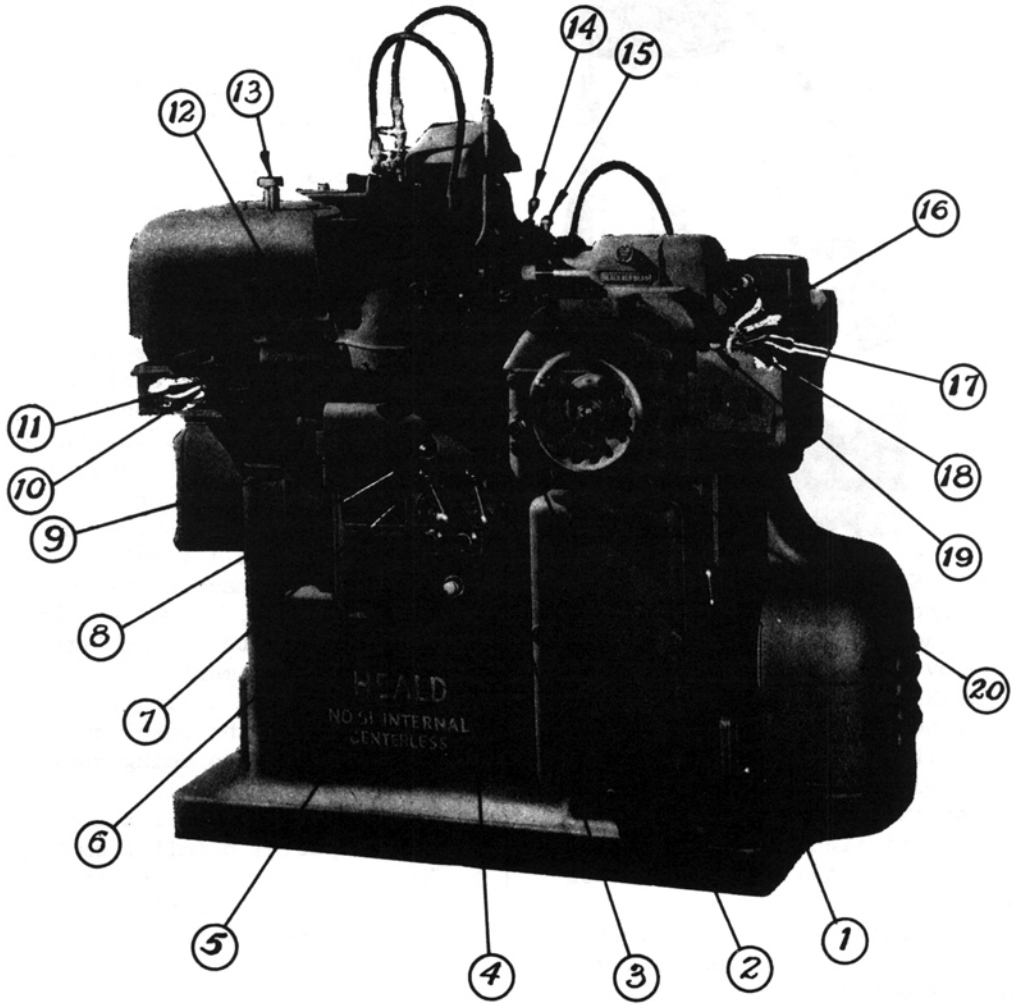


Fig. 802. Internal Centerless

1. Safety switch controlling all motors.
2. Wheel wear compensation knob.
3. Table traverse lever.
4. Knob controlling truing speed of table.
5. Delay valve controlling time of cross slide retraction.
6. Knob controlling finishing speed of table.
7. Throttle controlling table speed.
8. Hand lever for table runout.
9. Adjustment for amount of wheel retraction located here.
10. Trip screw for table runout when dressing wheel.
11. Trip screw for table runout when work is finished.
12. Traverse adjustment for regulating roll.
13. Knob for adjusting pressure roll.
14. Hand lever for operating wheel truing device.
15. Knob for micrometer adjustment of truing device.
16. Pick-up feed adjustment.
17. Lever to shut off all feed on wheel head cross slide.
18. Fine feed adjustment of wheel slide.
19. Coarse feed adjustment of wheel slide.
20. Handwheel for manual operation of wheel slide.

9. Why must the spindle bearings of the internal grinder be kept especially well lubricated?
10. In the grinding operation what is the relation of the grinding wheel with respect to the work?
11. Explain, exactly, how to chuck up a job in a four jaw chuck on the internal grinder.
12. Give six methods for holding work on the internal grinder besides the four jaw chuck and collets.
13. What care must be exercised in the chucking of thin walled bushings?
14. Explain exactly, the correct method of clamping work to a face plate on the internal grinder.
15. What are the most frequent causes of out-of-roundness of the work in internal grinding?
16. What two work operations is the operator of the internal grinder most frequently called upon to do?
17. What conditions must prevail for the production of internal cylinders on the internal grinder?
18. How can this condition be brought about?
19. How is extreme taper produced on the internal grinder?
20. How are slight tapers produced on the internal grinder?
21. Explain how to produce accurate taper work on the internal grinder.

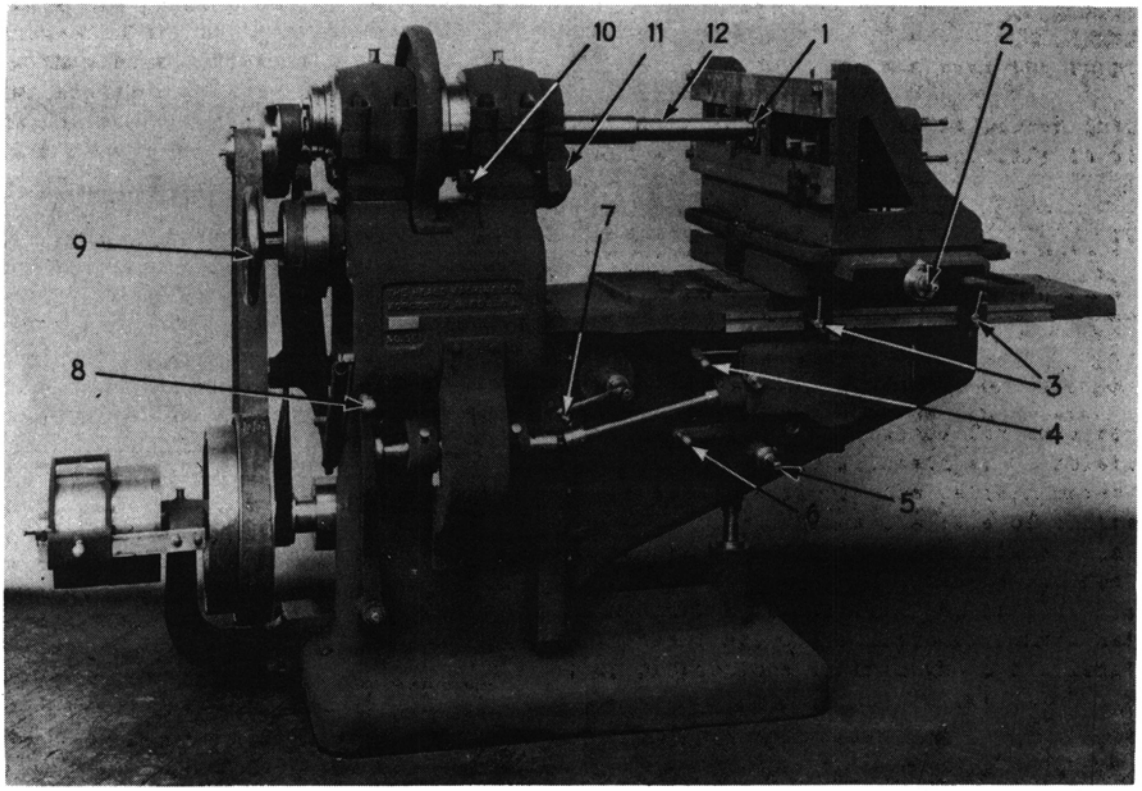


Fig. 803. Cylinder Grinding Machine

1. Grinding wheel.
2. Screw for moving table crosswise of saddle.
3. Adjustable stops for length and position of stroke.
4. Lever for controlling speed of table reciprocation.
5. Shaft for adjusting height of knee.
6. Lever for engaging table reciprocation.
7. Crank for hand control of table to or from the grinding wheel.
8. Lever for starting and stopping grinding wheel spindle.
9. Handwheel for turning spindle by hand.
10. Lever for adjusting the depth of cut in steps of .0005" on the diameter of the work while the spindle is rotating at high speed.
11. Hole sizing mechanism. Operated by turning a knob for small adjustments and by turning a crank when going from a small hole to a large hole.
12. Grinding wheel spindle. Comes in six different sizes for different work requirements.

22. Explain the use of the taper plug gage in producing an accurate taper.
23. What is meant by the term "freeze" when checking a hole with a plug gage?
24. How can this freezing be prevented?
25. Upon what two factors does the selection of the diameter of the grinding wheel for internal grinding depend?

TOOLS COMMONLY USED ON INTERNAL GRINDERS

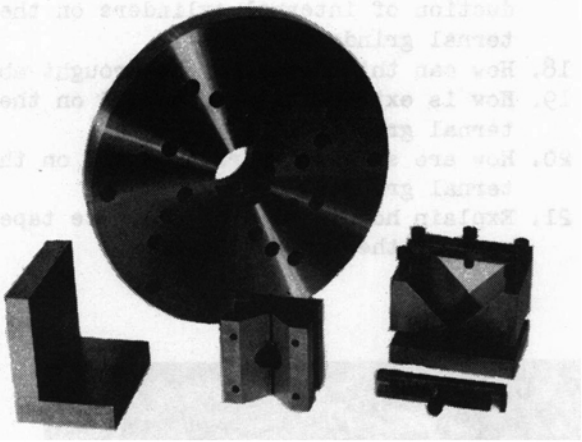


Fig. 804. 10' H & G face plate, 3" 90° Angle plate, 4" V-block.

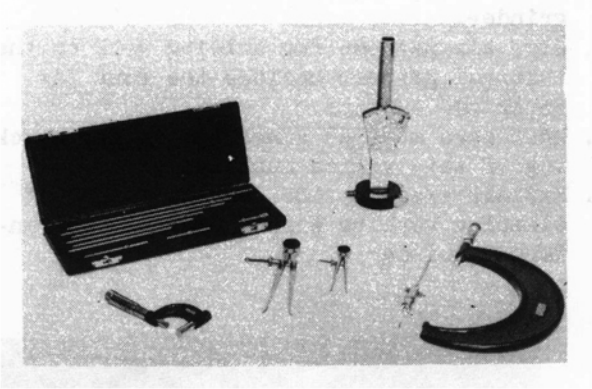


Fig. 805. Inside mics, Tube mics, Inside calipers, #3 Jo gun.

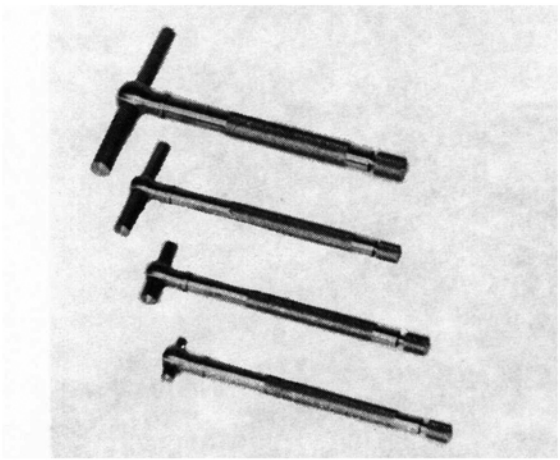


Fig. 806. Telescope Gages: AA, A, B, C.

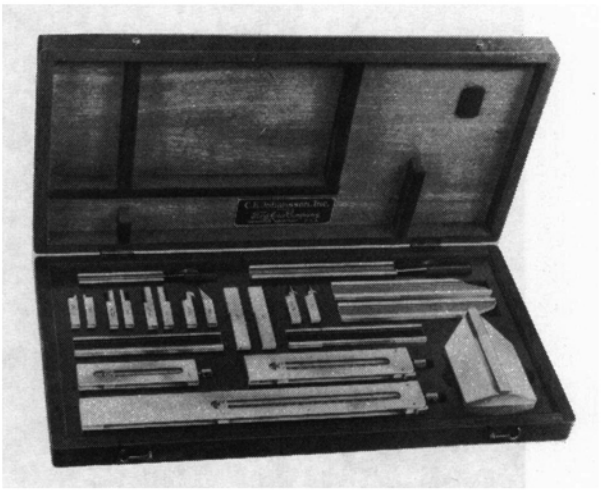


Fig. 807. Johansson Accessories

Fig. 808. Johansson Gage Blocks

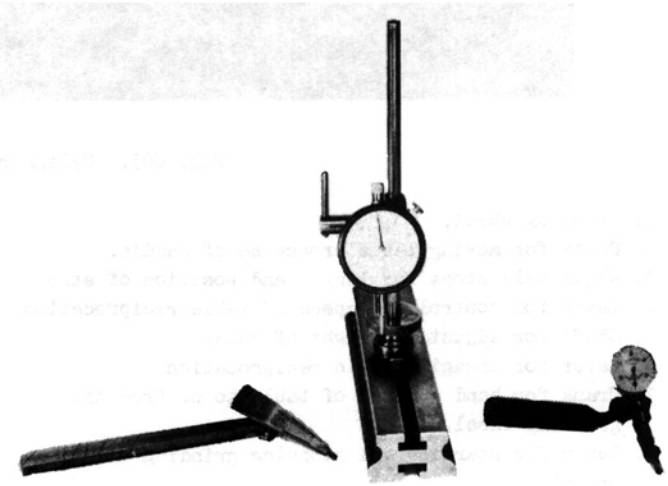


Fig. 809. Ideal Indicator, Dial Indicator, Last Word Indicator

26. What are the limits of speed in F.P.M. for internal grinding wheel?
27. What is meant by the term "bell-mouthing"?
28. What is the most frequent cause of bell-mouthing?
29. How can bell-mouthing be prevented?

COMMON SET-UPS AND OPERATIONS ON INTERNAL GRINDERS

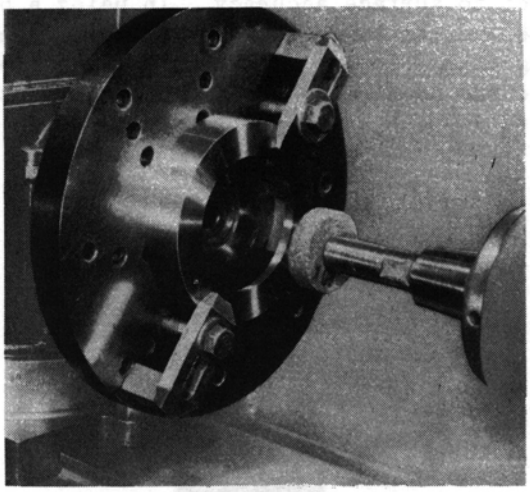


Fig. 810. Work Clamped to Face Plate

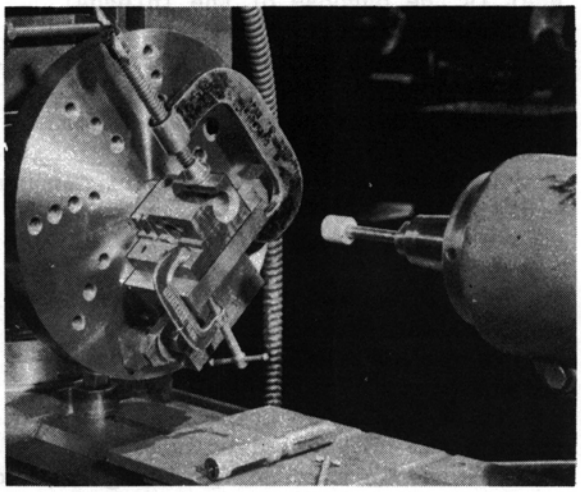


Fig. 811. Angle Plate Job on Face Plate

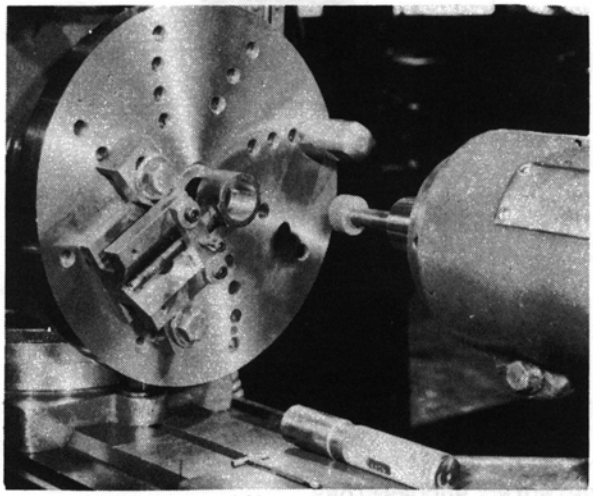


Fig. 812.. V-block Job on Face Plate

Fig. 813. Setting a 3" Jo Gun Using Accessories

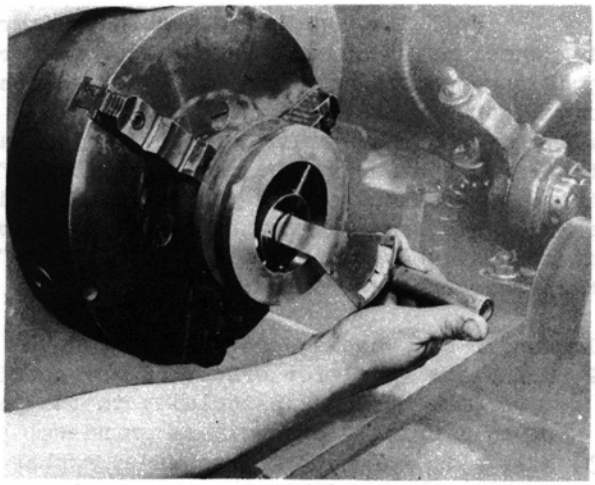


Fig. 814. Checking the ID of a Bushing with a 3" Jo Gun.

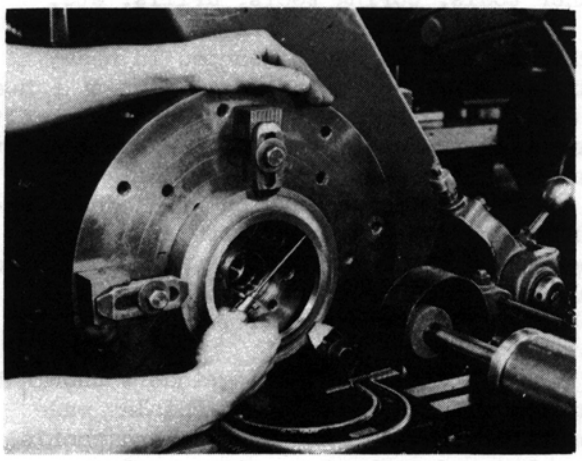


Fig. 815. Using Inside Mic and Extension Handle to Check ID of Bushing.

30. When checking work on the internal grinder how is the operator protected from the rotating grinding wheel?
31. What condition is necessary for the production of blind holes on the internal grinder?
32. What is the purpose of the Internal Centerless grinder?
33. How is the work held in an internal centerless?
34. For what kinds of work is the internal centerless suited?
35. By what name is the cylinder grinder commonly known?
36. What is the purpose of the cylinder grinder?
37. For what branches of industry is the cylinder grinder especially suited?
38. How is the work held in the cylinder grinder?
39. How is the cylinder grinder adjusted to produce the correct sized hole?
40. Wheels used in internal grinding are made from what kind of bond?
41. What grades of bond and abrasive sizes are commonly used in internal grinding wheels?
42. How is the operator of a tool room internal grinder protected from the rotating grinding wheel?
43. What care should be exercised in mounting an internal grinding wheel?
44. At what two times in particular should the operator of an internal grinder wear goggles?

MISCELLANEOUS GRINDERS

A Floor or Bench grinder (Fig. 816) is usually an electric motor mounted on a suitable base and having the rotor shaft extended from each side with a grinding wheel mounted on each end, for sharpening tool bits, planer tools, boring tools, drills, etc.

Some floor grinders for heavy duty work have a wheel shaft mounted in heavy bearings which are an integral part of a heavy cast base. The wheel is driven by a belt connecting the wheel shaft pulley to a motor mounted on the base of the machine.

For grinding cutting tools these machines should be equipped with fine grain wheels of a silicate or vitrified bond. For rough grinding, snagging or heavy work, the machine should be equipped with coarse free cutting wheels.

All floor or bench grinders should be equipped with adjustable eye shields made from fire glass or safety glass and should never be used without wearing goggles.

One end of a Floor or Bench grinder may be equipped with suitable sewed canvas forms, the periphery of which have been coated with an abrasive to be used as a buffer. Buffing or polishing jacks are used for the purpose of securing a good finish without respect to surface accuracy. In using a buffer care must be taken to keep the work from getting caught in the wheel and being pulled into the machine or serious injury might result.

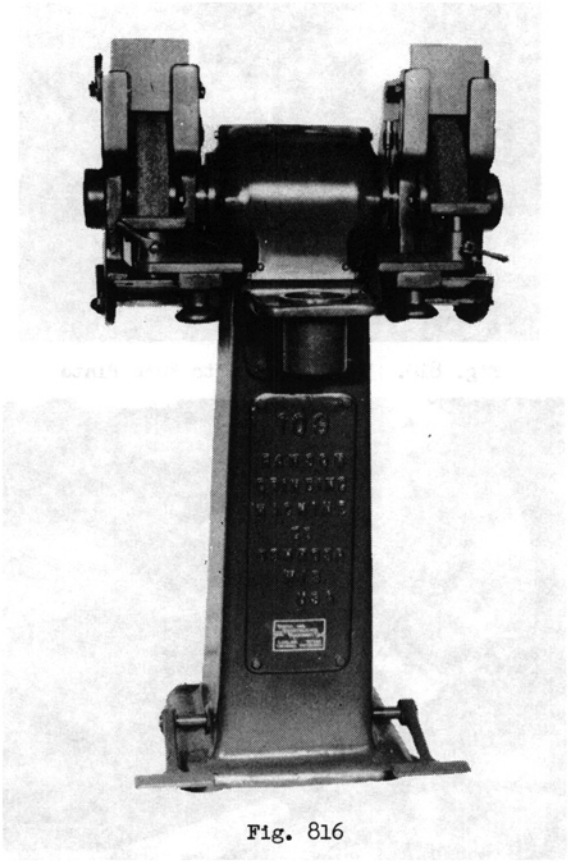


Fig. 816

GRINDING SUGGESTIONS

Well known grinding machine and grinding wheel manufacturers have made some very good suggestions relative to safety and economy of operation of grinding equipment. It will pay the student in grinding to pay particular attention to the more pertinent of these suggestions and if generally adopted should go a long way toward the elimination of machine abuses and accidents. A few of these suggestions follow:

1. Handle all wheels carefully.
2. Sound the wheel for cracks before putting it on the machine, especially in the case of cylindrical, and surface grinders.
3. Make sure the blotting paper or rubber gaskets are used between the wheel and wheel flanges.

4. Where a choice of machines is available use one that is rigid.
5. The wheel should fit freely on the spindle with about .003" to .005" play.
6. Use relieved or safety flanges which are at least one-third of the wheel diameter.
7. The spindle nut should be set up tight enough to properly hold the wheel.
8. Keep all wheel guards tight and in place.
9. After installing a wheel on a surface or cylindrical grinder turn it over by hand to make sure that it clears the guards and housing.
10. Keep the spindles and bearings well oiled to avoid overheating of the spindle and possible wheel breakage.
11. When starting to work in the morning, press the starter button on the machine and at the same time step to one side letting the machine run for at least one minute before engaging the wheel with the work.
12. When grinding long slender work on cylindrical grinders use steady rests to support the work and keep the shoes properly adjusted to the work. Make sure the driving pin for the grinder dog is firmly located in the arm on the drive plate.
13. When grinding wheels are removed from the machine handle them carefully and store them in a dry place.
14. When engaging the wheel with the work avoid excessive pressure; it is ruinous to the wheel, work supports, and spindle bearings.
15. In any grinding machine it is necessary for the work to be properly supported and that the operator keep his hands away from the revolving wheel and work.
16. When placing work on the magnetic chuck of surface grinders the table should be moved far enough away from the wheel so that the operator can locate the work without injury to his hands from the rotating wheel.
17. Since the table of a Horizontal Rotary grinder moves along its ways very easily, the work should be removed from the magnetic chuck by sliding it off in a direction perpendicular to the table traverse.
18. Close and easy fitting goggles should be worn on any dry grinding operation where there is danger of flying grit getting into the operator's eyes.
19. Exhaust hoods on dry grinders are provided for the benefit of the operator and should be kept in place and free from obstructions.
20. Magnetic chucks are expensive pieces of equipment. Keep them clean. Keep them free from scratches and nicks. Don't leave the machine for any length of time with the switch to the chuck turned on.

The thread grinder is a machine developed for the precision grinding of threads, and is becoming a necessity where accuracy and production are required or threaded work (both external Fig. 817 and internal Fig. 818) of various types and different materials.

The output is controlled by the automatic features of these machines. When it is necessary to grind only a few parts, the thread grinder can easily be set to new work and operated manually.

Besides greater accuracy, the thread grinder has other practical advantages. Exceptionally hard materials, including tough steel that is heat treated and are difficult to thread by other methods, can readily be threaded by these grinders. Threads are often ground directly from the solid. This

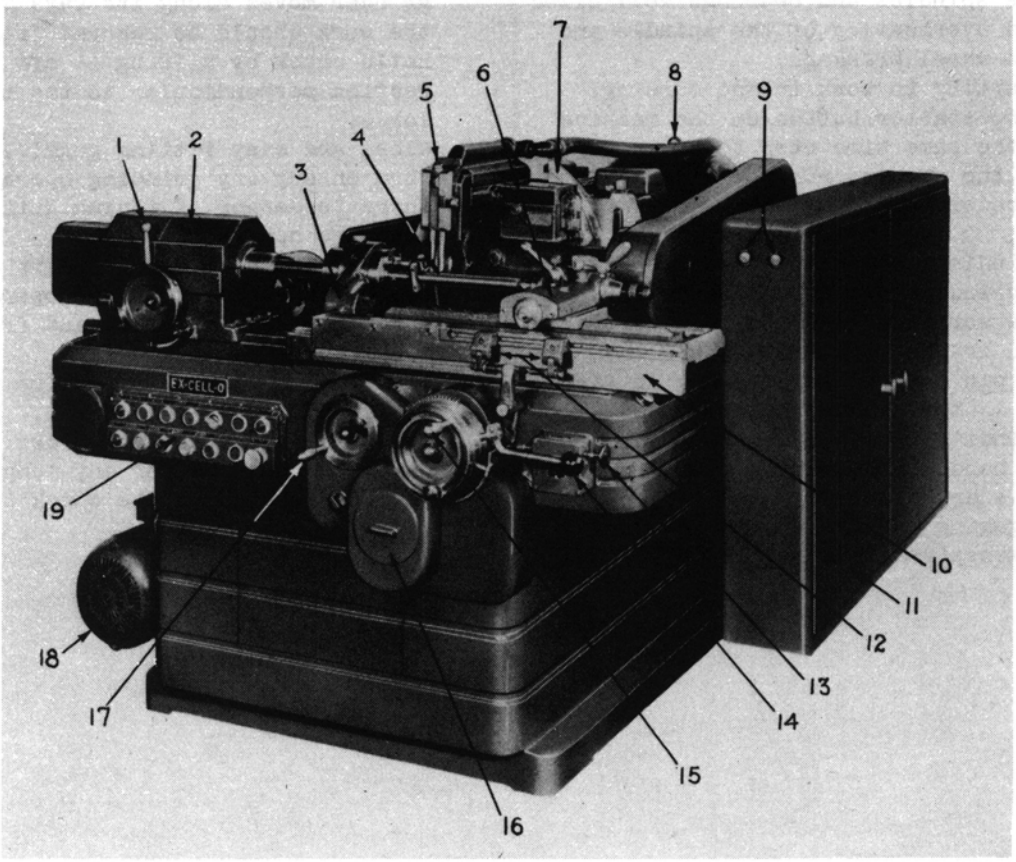


Fig. 817. External Precision Thread Grinder

1. Lead Pick-up
2. Work Drive and Lead Screw Housing
3. Work Head
4. Grinding Wheel
5. Coolant Valve
6. Tailstock
7. Helix Angle Graduation
8. Wheel Spindle Motor
9. Signal Lights Indicating Dressing of Wheel
10. Machine Table Slide
11. Electrical Compartment
12. Table Control Dogs
13. Manual Table Reverse Control Knob
14. Automatic Cycle Starting Lever
15. Size Setting Hand Wheel
16. Opening to Adjust Depth of Initial Grinding Cut
17. Manual Dresser Slide Adjustment
18. Work Drive Motor
19. Control Panel

naturally means a saving in time. Whether grinding fine threads from the solid or coarse threads previously roughed and given heat treatment, a tolerance on the pitch diameter of plus or minus .0002" can ordinarily be held.

The thread grinder uses a high precision ball bearing spindle to rotate the grinding wheel, the accuracy in size and finish of the ground piece depending to a considerable extent on the exactness with which the spindle operates.

On the external thread grinder shown in Fig. 817 for external threads the grinding wheel has a diameter of 18" and a thickness of 3/8", the internal thread grinder, Fig. 818, uses a grinding wheel from 3/4" to 4" in diameter. The grinding wheel is automatically kept dressed by the machine.

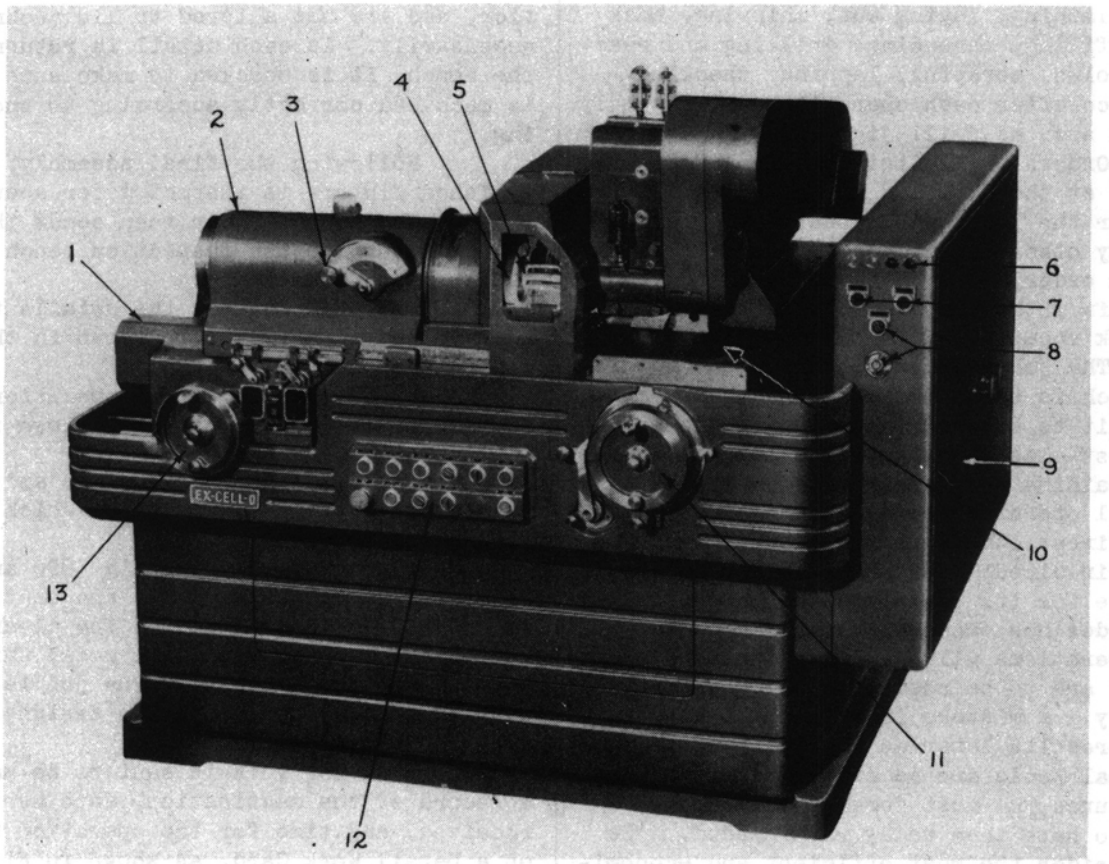


Fig. 818. Internal Precision Thread Grinder

1. Work Drive and Lead Screw Housing
2. Work Head Slide
3. Control for Right or Left-Hand Thread and Multiple Index
4. Work Piece
5. Grinding Wheel
6. Lights Indicating Cycle, Machine and Dresser Operation
7. Work Spindle Jog Control-Right and Left
8. Knob and Switch to Control Automatic Dressing
9. Electrical Compartment
10. Wheel Slide
11. Size Control Hand Wheel Assembly
12. Operator's Control Panel
13. Lead Pick-up and Automatic Backlash Compensation Control

Chapter 26

ROUTING OF BENCH TOOL WORK

The purpose of this group of sheets is to show the routing of Bench Work through the various departments of the Tool Room.

The term "Bench Work" is applied to the type of work done at the bench, consisting of planning, laying out, chipping, hack sawing, filing, threading, drilling and reaming of holes, scraping, lapping, checking each piece after each operation, and assembly of parts such as drill jigs, dies, gages, etc.

Orders coming into the Tool Room are received at the Clearing House. Here all the stock for the job, except that which is furnished by other departments and specified on the work order, is ordered on standard requisitions from the steel, tool, and miscellaneous stock rooms.

The job is then routed to the department which is to do the work and the foreman assigns it to a bench leader. A bench leader is a first-class toolmaker who has had sufficient training and experience to perform all the usual operations done in the tool room and to direct others in their performance. He is then in direct charge of the job and responsible for it. He plans the job carefully and decides how each detail will be made. Which operations will be done first and what machines are to be used must be considered carefully. A mistake in this might lead to serious results later as the job progresses. If special tools are necessary, such as jigs and fixtures, he must foresee this early enough to have them ready when needed. He must consider the most efficient and economical methods that he can use. A bench leader can often save many hours of time and a great deal of expense by having the work carefully planned, thus avoiding mistakes which might otherwise be made.

When the stock is received by the bench leader, it is checked with the blue print to make certain that the proper material and sizes have been sent. He then writes out a Tool Operation Card, routing the job to the required machine. This card is accompanied by a sketch or blue print showing the work that is to be done.

Since it is necessary for the bench leader to be able to locate any part of the job at any time, a complete record of each detail is kept on the back of the work order card. This record shows where each detail has been sent, the date sent, and the date returned.

He must follow up his orders so that the details are finished in the proper rotation, and are not allowed to lie around unnecessarily. As each detail is returned to the bench, it is checked to make sure that it is machined correctly according to the drawing.

Following the final assembly, the finished fixture is inspected for accuracy.

The bench leader then sends the finished fixture to the inspection bench for the final inspection.

The procedure of the details through the various departments is shown in the following sheets.

As an example of the operations and routing of a job, the expander shown in Fig. 819 is used.

The drafting room designs and makes a drawing of the expander, from which blue prints are made.

The order shown in Fig. 820 and the blue prints for the expander are sent from the work order department to the clearing house of the designated tool room, which in turn orders the material. The job is then sent to the bench foreman who assigns it to a bench leader.

When the work is sent to be machined, a record of the destination, date sent, date received, and time for the operation is kept on a Detail Work Card, as shown in Fig. 821.

A Tool Operation Card specifying the work to be done accompanies the stock and sketch or blue print to each machine. A new card is filled out for each new operation.

In this particular case, the Tool Operation Card is filled out as shown in Fig. 822 and sent with the blue print or sketch to the lathe.

After the job is returned from the Lathe Department, it is checked to determine whether the dimensions machined there are correct.

Details 1 and 2 are then sent to the Milling Machine Department, with a new Tool Operation Card specifying the operations to be done there.

After the milling is completed, both

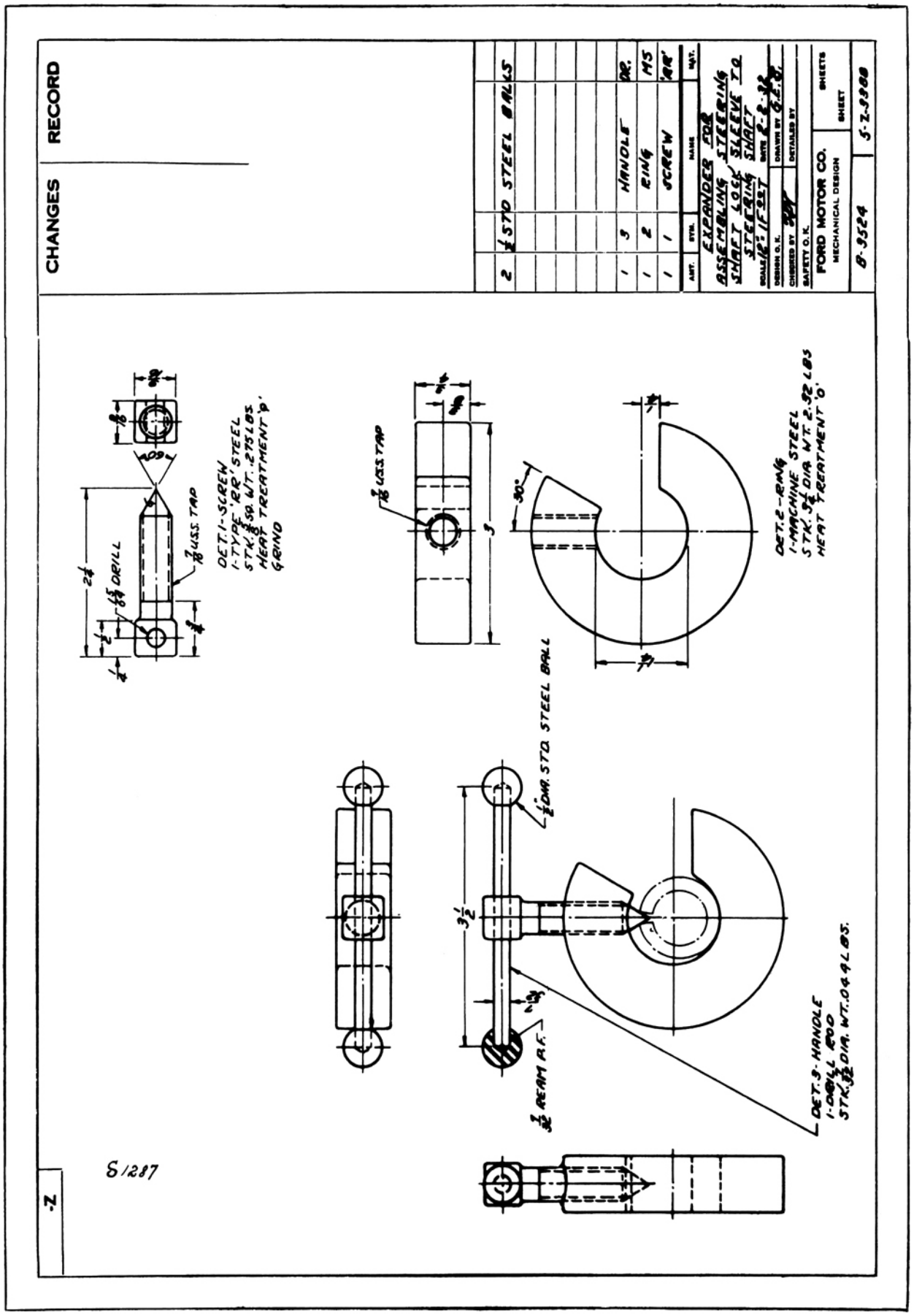


Fig. 819. Drawing of Expander

Ford Motor Company

SERIAL Nº 881507
5-Z-3588 PIOCH

ACCOUNT NO.	DEPT. CHG.	FACTORY WORK ORDER NO.	APPROVE
270	5042	S-48539	Le

ASSIGNED	CAST IRON	STEEL	NI CHROME	BRONZE	AL BRONZE	ALUMINUM	BRASS

Z-NUMBER	DETAIL	PART NO.	PATTERN NUMBER	PATT. LETTER
5-Z-3588		B-3524		

EXPANDER FOR ASSEMBLING STEERING SHAFT LOCK SLEEVE TO STEERING SHAFT.

TRADE SCHOOL: MAKE (6) COMPLETE.
DELIVER TO 5042 MOTORS BLDG

c.c.
N-650
HUSCHER

6
DEC. 15 1931

REASON NEW

DATE	REQUESTED BY	OF DEPT.	LOCATION	FOR DEPT.	LOCATION OF JOB	SUPT. O. K.
12-14-51	A.SMITH	SUPT.		5042	MOTORS BLDG	

Form 2063 3000M.—6-4-51

Fig. 820. Order Form

FORD FORM 2106 3000M 11-2-51

DETAIL WORK CARD

AMT.	PATT.	DETS.	K'D ST'K	STOCK REC.	STOCK REQ.			PLANER SHAPER OUT	PLANER SHAPER IN	LATHE OUT	LATHE IN	MILL OUT	MILL IN			BULLARD OUT	BULLARD IN	HEAT TREAT HARDENING OUT	HEAT TREAT HARDENING IN	GRINDER OUT	GRINDER IN			DEL.
6		1																						
6		2																						

Fig. 821. Work Card

details are again checked for correct dimensions.

The filing, drilling, tapping, and reaming operations are then completed at the bench, after which details 1 and 2 are sent to the Heat Treating Department to be hardened, tempered, and inspected.

They are then returned to the bench and detail 1 is sent to the Grinding Department.

When the grinding work is completed, detail 1 is returned to the bench, where it is again checked for size. The job is then ready for final assembly.

After the final assembly, the job is sent to the Inspection Department for final inspection, and from there to the clearing house for delivery to the department specified on the work order.

Form 3112 500M—4-15-36

Chg. Dept. 5042 **TOOL OPERATION CARD** Acct. No. 270

Order No. S-48539 | Part No. 3524 | Mach. No.

Z-No. 5-Z-3388 | Det. No. 1 and 2 | Date 2-22-38

Sent by Arnold | To Dept. Lathe

No. Pcs.	Patt. or Det.	REMARKS
6	1	Machine as per attached blue print.
6	2	

Foreman Stamptage Dept. T. S.

Inspected by Date

KEEP THIS CARD CLEAN AND RETURN WITH JOB

Fig. 822